This is a digital copy of a book that was preserved for generations on library shelves before it was carefully scanned by Google as part of a project to make the world's books discoverable online.

It has survived long enough for the copyright to expire and the book to enter the public domain. A public domain book is one that was never subject to copyright or whose legal copyright term has expired. Whether a book is in the public domain may vary country to country. Public domain books are our gateways to the past, representing a wealth of history, culture and knowledge that's often difficult to discover.

Marks, notations and other marginalia present in the original volume will appear in this file - a reminder of this book's long journey from the publisher to a library and finally to you.

Usage guidelines

Google is proud to partner with libraries to digitize public domain materials and make them widely accessible. Public domain books belong to the public and we are merely their custodians. Nevertheless, this work is expensive, so in order to keep providing this resource, we have taken steps to prevent abuse by commercial parties, including placing technical restrictions on automated querying.

We also ask that you:

+ *Make non-commercial use of the files* We designed Google Book Search for use by individuals, and we request that you use these files for personal, non-commercial purposes.

+ *Refrain from automated querying* Do not send automated queries of any sort to Google's system: If you are conducting research on machine translation, optical character recognition or other areas where access to a large amount of text is helpful, please contact us. We encourage the use of public domain materials for these purposes and may be able to help.

+ *Maintain attribution* The Google "watermark" you see on each file is essential for informing people about this project and helping them find additional materials through Google Book Search. Please do not remove it.

+ *Keep it legal* Whatever your use, remember that you are responsible for ensuring that what you are doing is legal. Do not assume that just because we believe a book is in the public domain for users in the United States, that the work is also in the public domain for users in other countries. Whether a book is still in copyright varies from country to country, and we can't offer guidance on whether any specific use of any specific book is allowed. Please do not assume that a book's appearance in Google Book Search means it can be used in any manner anywhere in the world. Copyright infringement liability can be quite severe.

About Google Book Search

Google's mission is to organize the world's information and to make it universally accessible and useful. Google Book Search helps readers discover the world's books while helping authors and publishers reach new audiences. You can search through the full text of this book on the web at `http://books.google.com/`

Library
of the
University of Wisconsin

Digitized by Google

Digitized by Google

Digitized by Google

PRINCIPLES OF MACHINE WORK.

Digitized by Google

By Robert Henry Smith

Massachusetts Institute of Technology

Text-Book of the Elements of Machine Work
192 pp., 5 x 8, 204 Illustrations.

Text-Book of the Principles of Machine Work
388 pp., 5 x 8, 434 Illustrations.

Text-Book of Advanced Machine Work
350 pp., 5 x 8, 400 Illustrations.

INDUSTRIAL EDUCATION BOOK COMPANY
BOSTON, U. S. A.

Digitized by Google

TEXT-BOOK OF THE PRINCIPLES OF MACHINE WORK

PREPARED FOR

STUDENTS IN TECHNICAL, MANUAL TRAINING, AND TRADE SCHOOLS, AND FOR THE APPRENTICE IN THE SHOP

ENGINE AND SPEED LATHES, DRILLING AND GRINDING MACHINES, CARBON AND HIGH-SPEED STEEL CUTTING TOOLS, MEASURING, TURNING, FITTING, THREADING, CHUCKING, DRILLING, REAMING, JIGS, FIXTURES AND CYLINDRICAL GRINDING

BY

ROBERT H. SMITH

MASSACHUSETTS INSTITUTE OF TECHNOLOGY

434 Illustrations

FIRST EDITION

INDUSTRIAL EDUCATION BOOK COMPANY

BOSTON, U.S.A.

Digitized by Google

Copyright, 1910,
BY
ROBERT H. SMITH.

Stanhope Press
F. H. GILSON COMPANY
BOSTON, U.S.A.

Digitized by Google

6172312

153063

TDH
SM.66
T.

PREFACE

In teaching mathematics, physics, chemistry, etc., text-books of classified information are available and are a required and necessary part of class-room and laboratory courses; thus the student advances rapidly and systematically, and the instructor is enabled to accomplish effective work.

In this the Age of Machinery, students, apprentices, and machine operators are handicapped by lack of text-books of classified information of the art and science of machine construction.

The aim of these books, — *Elements of Machine Work, Principles of Machine Work, and Advanced Machine Work* is to give the beginner the advantages of text-books as in the older subjects, that he may acquire the fundamental as well as advanced principles in a logical, systematic, and progressive manner and in the shortest time possible.

Machines, mechanisms, and tools are illustrated graphically by means of original perspective and mechanical drawings, and briefly and systematically described by condensed tables. Operations in machining, standard and typical problems in machine construction are given in condensed schedules which name the material, operations, machines, speeds, feeds, jigs, fixtures, and tools. Calculations are supplied by condensed rules and formulas. Facts and principles are supplied which a student or apprentice in school or shop must rediscover or obtain from instructor or foreman. As the subject is large and varied, it is impossible for instructor or foreman to do justice to it; consequently, the task is a difficult one and the beginner's progress extremely slow.

These books tell how to do things, with that theory which connects principles and practice, and no person can build or

Digitized by Google

superintend the construction of machinery without consciously or unconsciously understanding these problems and applying these principles.

To the manufacturers, teachers, associates, and other friends who have kindly assisted with information, help, and encouragement, I take this opportunity of expressing my indebtedness and appreciation.

R. H. S.

May, 1910.

Digitized by Google

CONTENTS

CHAPTER I.

PAGE

Engine Lathes, countershaft, line shaft, and belt connections described — Horizontal section of headstock — Transparent view of lathe apron, showing the three distinct feed mechanisms — Swing, length, and classes of lathes — Attachments for lathes — Engine lathe with rapid change-gear mechanism..... 1

Electrically-driven Machine Tools. — Electrically-driven engine lathe..... 9

Speed Lathes. — Countershaft, line shaft, and belt connection described — Lubricating bearings — Treatment when bearings rough up — Rust or corrosion — Changing speed belts of lathes — Warning against loose sleeves and careless actions near running belts, gears, milling cutters, etc..... 11

CHAPTER II.

Truing and Alining Centers. — Center gage — Requirements for successful use of engine lathe — Truing engine lathe centers with center truing tool — Grinding centers — Truing speed lathe centers — Setting dead center in alinement for straight turning. 15

Center Holes in Lathe Work. — Table of center-hole dimensions — Countersinks..... 19

Hand and Machine Methods of Centering. — Straightening shafting, rods, and bolts..... 23

CHAPTER III.

Cutting Tools. — Rake, clearance, and cutting angle defined. — Rough and finish turning..... 27

Lathe Cutting Tools for Cast Iron, with Chart. — Right and left tools — Angles of lathe-cutting tools — Height of tools for cutting operations — Rough squaring and turning with round-nose tools — Rough and finish squaring with side tools — Rough and finish turning with diamond-point tool — Square-nose finishing tool..... 28

Digitized by Google

PAGE

Grinding Lathe Tools. — Grinding round-nose, side, diamond-point, square-nose, and threading tools — Grinding high-speed cutters and various hand tools, on grindstone or emery wheel — Universal tool grinder for duplicating shapes and angles of lathe and planer tools — Oilstoning tools 37

CHAPTER IV.

Setting and Using Outside Calipers. — Adjusting tool to turn to desired diameter — Transferring settings from one pair of calipers to another 45

Cutting Speeds, Cut Meter, and Feeds. — Cut Meter — Attachment for speed indicator for obtaining surface speed — Rules for obtaining cutting speeds — Table of cutting speeds for various metals, including high-speed steels — Cutting feeds 47

Lubricants for Cutting Tools. — Table showing when to use a lubricant and when to machine dry 50

Inspecting and Measuring Stock, Oiling and Cleaning Machines. — Care of machines, tools, and benches 52

CHAPTER V.

Engine Lathe Work. — Schedule drawings of machine parts — How to begin lathe work — Mounting work on centers — Turning work from end to end 54

Centering, Squaring, and Straight Turning, with schedule of operations 57

Filing Lathe Work. — Mill files — Speed for filing — Round and half-round files for inside rounds and fillets 59

Micrometer Calipers. — Micrometer principle of measuring — Measuring the work and reading the micrometer — Methods of holding micrometer to measure work held in the hand, lathe, or on the bench — Large micrometers — Table of decimal equivalents of common fractions 61

Vernier Calipers. — Vernier principle of measuring — Measuring work and reading the vernier — Making inside measurements with vernier — A ten-thousandth micrometer with vernier on the barrel 65

CHAPTER VI.

Fits in Machine Construction, with Tables of Allowances. — Typical examples of fits — Materials used for different kinds of bearings — Classes of fits — Allowances and limits for running, driving, and forcing fits — Taper forcing fits — Pressure and allowances for forcing fits 68

Digitized by Google

PAGE

Forcing Press. — Examples of forcing and shrinking fits, with tables of allowances and limits — Holes and hole gages — Turning and filing fits — Grinding fits 72

Standard and Limit Gages. — Ring, plug, and caliper gages and reference disks — Standard end measuring rods and limit gages — Special gages 77

CHAPTER VII.

Tapers in Machine Construction. — How expressed — Standard and special — Method of turning — Calculating distance to set over footstock — Using a pattern to obtain set-over 80

Taper Turning and Fitting. — Turning, filing, and fitting a Morse taper, with schedule of operations 82

Taper Attachment for turning tapers in an engine lathe. — Turning a taper on a large drill socket 84

Straight Turning and Fitting. — To turn and file a straight running fit, with schedule of operations 85

CHAPTER VIII.

Lathe Tools for Steel or Wrought Iron. — Side and diamond-point tools — Roughing tools — Finishing tools — Spring and shear tools — Left side and left diamond-point tools — Half diamond-point tools 87

Holders and Cutters. — Straight holders and cutters — Useful forms of cutters — Chart of lathe tool holders for squaring, turning, boring, threading, and forming — Bent holders — Double holders 93

Cutting-Off Tools. — Cutting off stock with forged tool — Cutting-off tool holder and cutter 97

Turning Steel. — Preparing shaft blanks, with multiple schedule of operations 98

CHAPTER IX.

Screw Threads. — Right and left threads — Forms of threads — Single and multiple threaded screws — Pitch and lead of thread — Threads per inch — Measuring threaded work — Counting threads — The Sharp V thread — The United States Standard thread 102

Threading Tools for United States Standard and Sharp V threads — Setting tool 107

Threading or Screw Cutting. — Calculating simple gearing — Description of screw-cutting mechanism — To set up lathe for screw cutting and cutting the thread, with schedule of operations —

Digitized by Google

PAGE

Fitting thread to nut — To reset tool to resume cut after regrinding — To cut left threads — To thread to a shoulder — Fractional threads and methods of calculating gearing — Compound gearing — To calculate gearing for a given lead — To calculate gearing for metric screw threads with English lead screws — Translating gears — Metric lead screw — Catching the thread or threading long screws without backing belt — Threading taper work — Whitworth (English) standard threads 110

Bolt and Nut Making. — Squaring and turning bolts. — Nut mandrels — Squaring and chamfering nuts, with schedule of operations — Making clamp nut — To make finished bolt, with schedule of operations 121

CHAPTER X.

Chucks. — Independent, universal, combination, drill, draw-in, and special chucks 126

Face Plates. — Holding work on face plate — Balancing work with counterweights — Holding work with angle plate on face plate — Lathe axis indicator 131

Chucking in engine lathe, with schedule of operations. — Chucking with flat drills and reamers — Chucking with drill holder and steady rest 134

Reaming. — Classes of reamers — Irregularly spaced teeth — Reamers for brass — Hand reaming in vise — Reaming in vertical drilling machine and engine lathe, with schedule of operations — Adjustable reamers — Reaming stands — Fluted chucking reamers — Power reaming in engine lathe, with schedule of operations — Rose chucking reamer — Fluted and broach reamers for taper pins — Taper reaming in speed lathe 139

CHAPTER XI.

Mandrels or Arbors solid and expanding — Bridges in hollow castings — Revolving dead center for pipe turning — Special mandrels — Driving or pressing mandrels in or out of work — Mandrel or arbor block — Mandrel or arbor press 146

Mandrel Making. — Making a standard lathe mandrel, with schedule of operations 149

Turning and Finishing Flanges. — Rough and finish facing and turning — Making a cast-iron flange, with schedule of operations 152

Turning and Finishing Pulleys. — Tapering or crowning face — Making a pulley, with schedule of operations — Locating set screws 156

Digitized by Google

PAGE

Polishing Lathe Work. — Abrasives, speeds, and machines used — Order of applying different numbers of emery cloth — Polishing flanges and shafts — Polishing brass and copper 160

CHAPTER XII.

Hand Turning. — Cutting speeds — Methods of holding work in speed lathe — To turn and square with graver and round-nose tools..... 164

Making Machine Handles, with schedule of operations — Templets as guides to uniform production..... 167

Drilling, Tapping, and Hand Threading in Speed Lathe. — Chucking with twist drill in speed lathe — Tapping and threading work in speed lathe — Hand chasers to thread screws and nuts, with method of operation — Using the ripper and planisher for brass finishing — Filing brass — Burnishers and burnishing metal — Hand cutting-off tools — Inside hand turning and boring — Slide rest for speed lathes..... 171

Spring Winding. — Coil springs — Winding a compression spring with a spring winder — To gear a lathe to wind a spring — Winding a compression spring with a wire hook..... 179

CHAPTER XIII.

Brass Finishing. — Turning brass in engine lathe — Use of round-nose and front tools — Binding post and nuts, with schedule of operations..... 182

Machining Bronze, Copper, etc. — Aluminium — Babbitt — Lead — Vulcanite, or hard rubber — Fiber — Rawhide 187

Nurling. — Hand and machine nurling, with schedule of operations 188

Curved Turning and Forming tools for engine lathe work — Holders and cutters..... 190

Steady and Follower Rests. — To turn slender shaft with steady rest set to spot, with schedule of operations — To turn slender shaft supported by steady rest set to cat head — To turn slender shaft supported by follower rest..... 191

CHAPTER XIV.

Cylindrical Grinding Machines. — Machine grinding — The principle of grinding on two dead centers — Classes of grinding machines — Universal grinding machine, with schedule of parts..... 195

Grinding Wheels. — Abrasives used in their composition — Chart of wheels — Vitrified, silicate, elastic, tanite, vulcanite — Celluloid, and combination grinding wheels — Shapes of wheels —

Digitized by Google

PAGE

Mounting wheels — Table of speeds — Direction of rotation of wheel and work — Feed and depth of cut — Truing wheels — Truing centers — Methods of grinding work — Grinding work straight — Grinding work taper — Wet and dry grinding — Lubricants for grinding — Allowances for grinding — Rough and finish grinding — Expansion of work — Seasoning — Measuring tools 198

Problems in Cylindrical Grinding. — Grinding running fit in universal grinding machine, with schedule of operations — Grinding forcing fit in universal and tool-grinding machine — Grinding standard mandrel in universal grinding machine — Grinding to a shoulder — Commercial grinding — Grinding a cast-iron roll in plain grinding machine — Grinding a shaft in plain grinding machine, with use of back rests — Duplicate grinding in the automatic magnetic sizing grinder — Grinding taper collet — Grinding a phosphor-bronze taper bushing..... 209

CHAPTER XV.

Drilling Machines. — Vertical, horizontal, multiple-spindle, and radial..... 222

Vertical Drilling Machine, with schedule of parts..... 222

Twist Drills of carbon and high-speed steels — Shape and angle of point — Good and bad grinding — Lip clearance — Thinning the point..... 225

Grinding Drills. — Drill grinder — Machine and hand method of grinding — Drilling cast iron and steel — Using high-speed steel drill twisted from flat bar — Flat drills — To grind a twist drill for brass — Straight-groove drill for composition, brass or sheet metal..... 227

Sizes of Drills designated by common fractions, letters, and numbers, with decimal equivalents, gages, and tables — Body, reamer, and tap drills..... 234

Speeds and Feeds of Drills. — To calculate speeds and feeds of drills — Table of carbon-steel drill speeds and feeds — Tables of spindle speeds for belt-driven vertical drilling machines and speed lathes — Hand and power feeds — Tables of spindle and gear speeds for high-speed drilling machine..... 237

Laying Out and Drilling Holes. — Spring dividers — Rough, approximate, and accurate drilling — Laying out the work — Inspection circle — Drawing the drill, with schedule of operations — Accurate drilling, reaming, and tapping holes, with schedule of operations — Drilling for set screws — Allowances for bolt holes, cap screws, and stud bolts — Three methods of drilling in alinement without a jig — To drill work held by hand or in a vise on

Digitized by Google

PAGE

the drilling table — Drilling work held against an angle plate — Use of three and four-groove twist drills — Drilling diametrically through a shaft — Automatic drill chuck and collet — Cutting off by drilling — Slotting by drilling, plugging, and drilling — Lead holes for large drills — Starting a drill on a slanting surface — Drilling part of a hole — Cutting boiler-tube holes — Drill extension — Leveling work on drilling table — Drilling in speed lathe 242

Deep Drilling. — Oil drills — Vertical drilling with oil drill — Horizontal drilling with hollow oil drill and extension 257

CHAPTER XVI.

Drilling Jigs, and Multiple-Spindle Drilling Machines.—Interchangeable machine parts — Classes of drilling jigs — Improvised jigs — Plate jigs — Drilling and tapping engine cylinder heads — Multiple-spindle drilling machines — Box jigs — Drilling, reaming, and tapping in different directions with box jig 260

Radial Drilling Machines. — Automatic tapping attachment — Plain and universal radial drilling machines — Jig vise — Drilling and counterboring duplicate parts 265

Taps and Dies. — Sizes of taps — Hand, nut, pulley, hob, or master, and adjustable taps — Tap wrenches — Tap drill holes — Loose nuts — Hand tapping a rough nut, with schedule of operations — Accurate hand tapping — Tapping in vertical drilling machine, with schedule of operations — Jig tapping — Tapping *through* holes and *bottom* holes — Removing broken taps — Power tapping — Tap drills — To calculate root diameter for both United States Standard and Sharp V threads, with tables — Hand dies — Automatic dies — Power threading — Hand threading a bolt, with schedule of operations 269

CHAPTER XVII.

Bolts and Nuts. — Styles of heads — Classes of bolts: rough, semi-finished, and finished — Machine, planer, stud, improvised — Expansion, square, and hexagonal bolts — Square, hexagonal, check, and castle nuts 282

Cap and Set Screws. — Cap screws, collar screws, and tap bolts — Dowel pins — Lag screws, hanger bolts, and wood screws — Set screws — Thumb-screws and nuts 287

Machine Screws. — Numbered sizes — American Society of Mechanical Engineers' Standard, and American Screw Company's Standard, with tables — Tapping with small taps — Threading with small dies — Machine screw and wire gages — Tables of tap drills 292

Digitized by Google

PAGE

Counterboring and Countersinking. — Solid and adjustable counterbores — Countersinks 299

Calculating Diameter of Blank to Mill Square or Hexagonal. — Diameter of blank to mill or file square or hexagonal 303

Indexing in Engine Lathe. — Dividing the circumference of work into equidistant parts for drilling, filing, etc. 304

CHAPTER XVIII.

Inside Calipers and Inside Micrometers. — Measuring diameter of holes — Adjusting tool to bore hole to setting of inside calipers — Inside micrometer calipers, and method of using and reading ... 305

Boring and Inside Threading. — Setting and using boring tools in lathe — Holders and cutters — Inside threading tools for United States Standard or Sharp V threads — Setting inside threading tools — Schedule of operations for inside threading — Finishing inside thread with tap — Interrupted thread taps — Cutting an inside thread to a shoulder 308

Square Threads. — Square threading tools — Inclination — Holders and cutters — Setting square threading tools — Square thread taps — Method of cutting a square thread screw, with schedule of operations — Making the nut and fitting the screw 314

Acme Standard or 29° Threads. — Making the tool — Setting 29° threading tools — 29° thread taps — Cutting a 29° thread screw, with schedule of operations — Making the nut and threading the screw 325

Multiple Threads. — Cutting double square threads, with schedule of operations — Multiple threading tools 335

Alinement Drilling and Tapping. — Fixed nuts — Drilling and tapping cross-feed screw nut in axial alinement, with schedule of operations 337

Eccentric Turning. — Laying out and turning an eccentric shaft, with schedule of operations — Making an eccentric mandrel — Turning an engine eccentric, with schedule of operations — Laying two throw 90° crank-shaft center fixtures — Alining center fixtures, and laying out cranks — Turning crank shaft 338

CHAPTER XIX.

Tables and Other Data Used in Machine Work. — Electric units — Tables and formulas for United States Standard bolts, nuts and automobile screws and nuts — Shrink rules for various metals — Tables of Brown & Sharpe, Morse, and Jarno tapers — Tables of sheet metal, plate, and wire gages, and their uses 344

Digitized by Google

M and *M'* — Oil holes (oiled before using back gears).

N and *N'* — Oil holes (oiled before using back gears).

To Operate Back Gears.

For direct cone drive, slide nut *F* is in slot in cone *A*, back gears "out." To use back gears, drop bolt *E* and secure, throw lever *K* forward. To obtain direct cone speed again, throw lever back, loosen *E* and turn lathe until *F* engages slot in *A*. Tighten *E*.

Double and Triple Back Gears.

To obtain a greater reduction of speed, lathes are built double or triple back geared.

Attention. — Gear guards are often provided to prevent accident and to keep dirt and chips from gear teeth.

10. A typical lathe apron. Fig. 3. — The apron of a lathe carries the greater part of the feed mechanism. Fig. 3 shows a lathe apron and the three distinct mechanisms, the long. feed, cross feed, and screw-cutting feed. The first is used for moving carriage back and forth along bed for turning; the second for moving cross slide in and out for squaring; the third, the lead screw and split nut, for moving carriage along bed for cutting screw threads.

To change direction of feeds. — The reversing mechanism to change direction of rotation of feed shaft may be in the headstock and operated by lever 27, Fig. 1, or in the lathe apron.

LONG. FEED. CROSS FEED. LEAD SCREW.

11. *Long. Feed.*

A — Long. feed handle.

B — Pinion.

C — Spur gear.

D — Sliding stud.

E — Sliding pinion, on inner end of stud *D*.

F — Feed rack, fastened to inner side of bed.

G — Splined feed shaft.

H — Feather-keyed worm held in bracket *H'*.

J — Worm gear.

K — Friction clutch to connect *J* and *L*.

L — Pinion fast to *K*.

M — Knob controlling clutch *K*. Hand long. feed is obtained by rotating *A* which drives through *B*, *C*, *E* to *F*. Power long. feed is obtained from *G*. Clutch *K*, shown out of action, is thrown in, which causes *G* to drive through *H*, *J*, *L*, *C*, *E* to *F*.

Digitized by Google

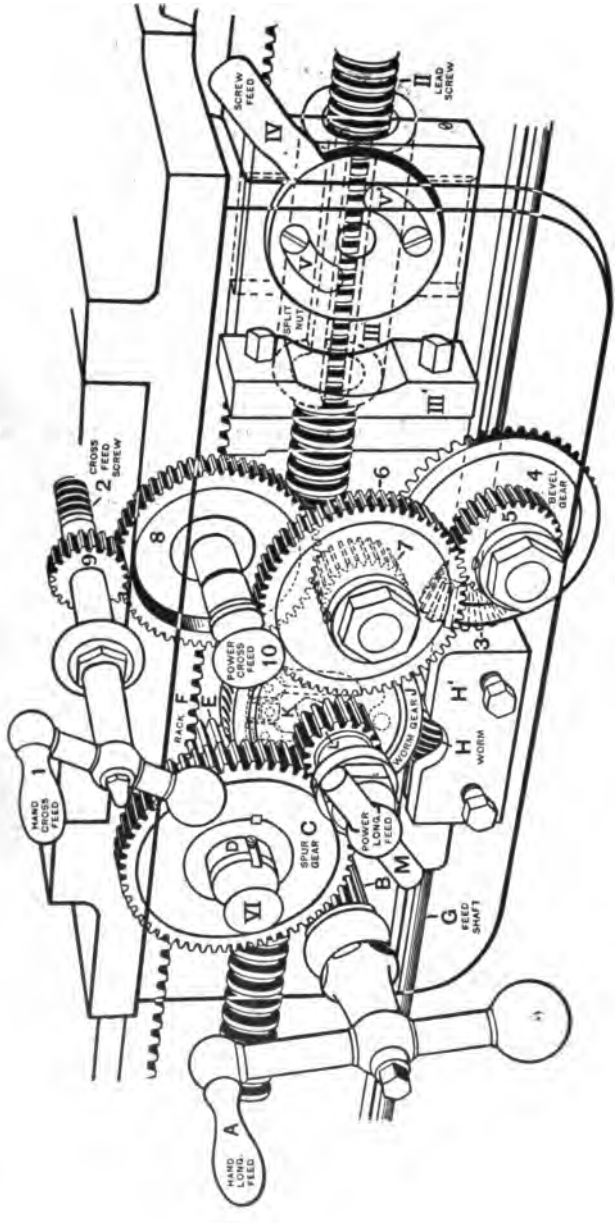


FIG. 3.—LATHE APRON SHOWN TRANSPARENTLY.

Digitized by Google

12. *Cross Feed.*

1 — Cross-feed handle.
2 — Operating screw, in nut under cross slide.
3 — Bevel pinion feather-keyed to *G*.
4 — Bevel gear.
5 — Pinion fast to 4.
6 — Driving gear.
7 — Pinion fast to 6.
8 — Gear always in mesh with 7.
9 — Cross-feed pinion.
10 — Knob controlling position of 8. Hand cross feed is obtained by rotating handle 1. Power cross feed is obtained by meshing 8 with 9 by means of knob 10.

13. *Lead Screw and Split Nut.*

II — Lead screw.
III — Split nut.
III′ — Split nut bracket.
IV — Lever to operate split nut III.
V and V′ — Cams closing split nut III.
VI — Knob to disengage *E* and *F* when screw cutting. To operate for screw cutting, pull out knob VI, throw IV downward, closing split nut III.

There are types of lathes, where a splined lead screw performs the combined duty of feed shaft and lead screw, the worm and bevel pinion being driven directly by it.

Attention. — Care should be taken not to have both long. feed and lead screw thrown in at the same time. Some lathes are fitted with devices which will prevent this.

14. Swing of lathes.—A lathe is designated by its swing and total length of bed. A 14″ × 6′ engine lathe will swing fourteen inches in diameter over ways, but will only swing about 8″ over a rise and fall rest, and about 10″ over a plain or compound rest. In length it will turn six feet less the combined length of head and footstock. A 6′ bed will turn about 42″ between centers.

15. Classes of lathes. — Lathes are divided into many classes, some of which are designed especially for the work performed upon them, as the wheel lathe, axle lathe, pulley lathe, turret lathe, bench lathe, jeweler's lathe, etc., and for ordinary work the engine lathe, which when supplied with special attachments is called a tool-maker's lathe.

Digitized by Google

16. Attachments for lathes. — Among the attachments for lathes are the taper attachment, compound rest, steady rest, follower rest, and attachments for milling and grinding.

17. Stops for duplicating sizes. — In addition to the thread stop some lathes are equipped with long. and cross-feed stops. After the first piece is turned or threaded to size, the back stop is set to check movement of cross slide. The carriage or long. stop is used for shoulders and lengths. By aid of these stops a number of pieces can be duplicated.

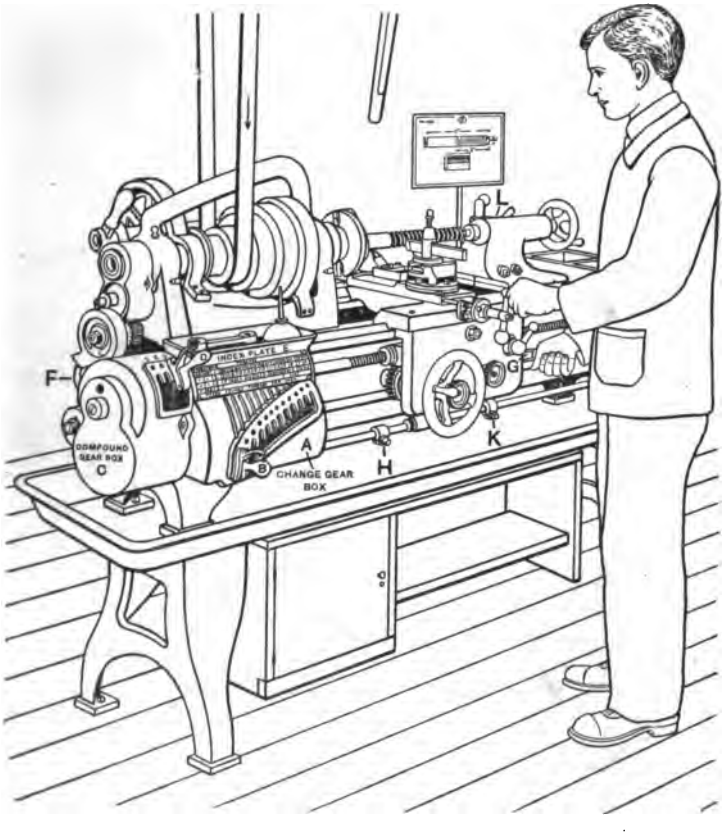


FIG. 4.—ENGINE LATHE WITH A RAPID CHANGE-GEAR MECHANISM.

Digitized by Google

18. Lathe with a rapid change-gear mechanism for threads and feeds. — Some lathes, as in Fig. 4, are equipped with a rapid system of change gears by which different threads or feeds are obtained quickly.

ESSENTIAL PARTS.

A — Change gears in cone form, on end of lead screw.

B — Handle operating change gears.

C — Compound gears.

D — Handle operating compound gears.

E — Index plate, giving positions for handles *B* and *D* to cut a desired thread.

F — Sector to carry gear for cutting special thread.

G — Lever to reverse carriage.

H and *K* — Automatic carriage stops. Carriage stop (invisible) is located on back ways under letter *L*.

19. Cutting a screw using a rapid system of change gears. — To cut a screw of five threads per inch, find 5 on index *E* and place handle *B* in notch and hole under it; place handle *D* under hole 3 on compound gear box, which is indicated in the third column on the same line as 5.

For threads given on the index, no change of gears is necessary. Gears may be calculated for other threads and applied as on the ordinary lathe. The feed for turning is seven times threads per inch expressed in turns per inch of tool travel.

20. For threading short screws the carriage may be reversed by moving reversing lever *G* up or down, depending on whether a right or left thread is being cut; or the automatic stops *H* and *K* may be used.

ELECTRICALLY-DRIVEN MACHINE TOOLS.

21. Arrangement of machine tools for electric drive. — They may be group-driven by electricity by using a constant-speed motor to drive group line shaft, or individually driven by attaching a constant or variable-speed motor to each machine. Constant-speed motors are used on machines that need but little speed variation, or on machines that have a large variety of mechanical speed changes.

Digitized by Google

22. Electrically-driven engine lathe — Variable-speed motor. *A*, Fig. 5, gives a wide speed variation. Speed is electrically controlled from apron. To start lathe, move handle *B* in direction of arrow. Controller *C* is provided with graduated wheel *D*. When a desired speed is obtained, pin *E* is

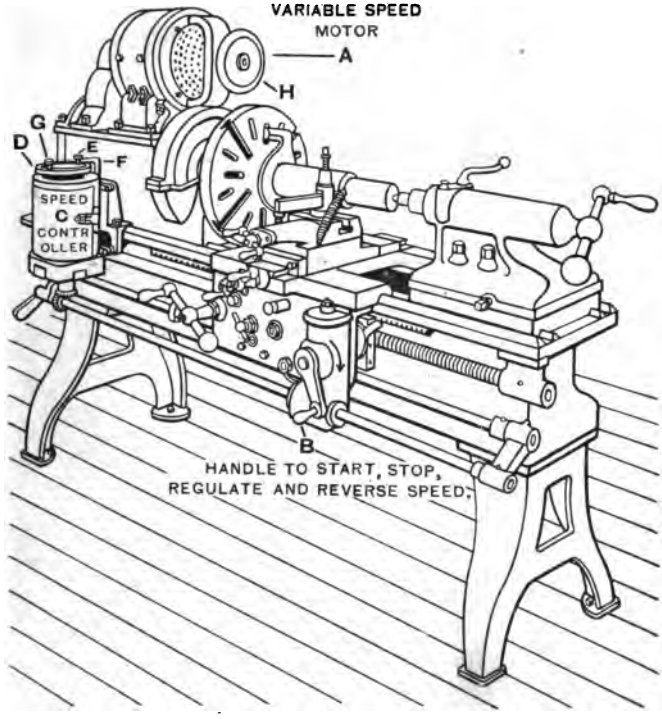


FIG. 5.—ELECTRICALLY-DRIVEN ENGINE LATHE.

placed in wheel *D* so that it will come against stop *F*. By this arrangement any speed can be duplicated. Pin *G* prevents lathe from being reversed. When it is desired to reverse lathe, remove pin *G* and move handle *B* in opposite direction. Wheel *H* is used to revolve lathe spindle by hand.

Digitized by Google

SPEED LATHES.

23. Speed lathe, Fig. 6 (often called a hand lathe), on account of its high speed and simplicity has a wide use in machine construction for hand turning, drilling, chucking, filing, polishing, etc.

CONE HEADSTOCK. COUNTERSHAFT DRIVE.

Essential Parts.

24. *A* — *Bed.*

B and *B'* — Legs fastened to bed and floor.

C and *C'* — *V* ways planed and scraped.

D — Headstock bolted to ways.

E — Footstock, position adjustable.

F — Clamp to fasten footstock to ways.

G — Spindle.

H — Handle operating footstock spindle.

I — Binder for clamping *G*.

J — Dead center.

K — Oil well and oiler for dead center.

L — Tee rest.

M — Tee-rest holder.

N — Screw to hold *M* in place.

O — Slide.

P — Lever to clamp *O* where desired.

25. *Headstock.*

1, 2, 3, 4 — Steps on headstock cone.

5 — Thrust bearing and end adjustment.

6 — Face plate, slotted to receive dog.

7 — Live center.

8 and 8' — Oil holes for live spindle.

26. *Countershaft (tight and loose pulley type) and Line Shaft.*

9 — Speed belt.

10 — Countershaft cone pulley.

11 — Headstock cone pulley.

12 — Countershaft mechanism (consists of shaft cone pulley, tight and loose pulley, and belt-shifting mechanism).

13 and 13' — Hangers (bolted to hanger plank).

14 — Hanger plank.

15 — Driving belt.

16 — Wide straight-faced pulley.

17 — Line shaft.

18 — Line-shaft hanger.

19 — Hanger plank.

20 — Shipper rod carrying two belt fingers.

21 and 21' — Belt fingers (21' not visible).

22 — Shipper pole pivoted to 14 and to 20. Belt fingers 21 and 21', between which belt 15 passes, are moved by 20 by means of 22, shifting belt from loose to tight pulley.

23 — Loose pulley.

24 — Tight pulley.

Digitized by Google

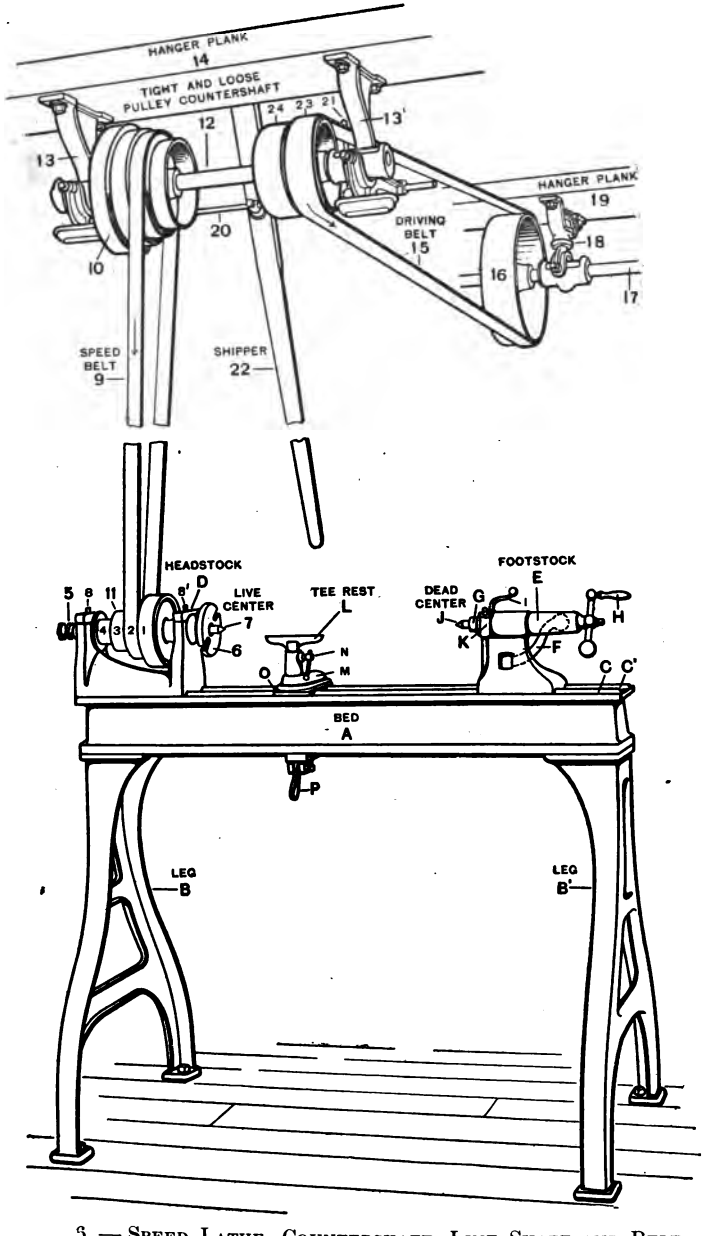


FIG. — SPEED LATHE, COUNTERSHAFT, LINE SHAFT AND BELT CONNECTIONS.

Digitized by Google

27. When and how to oil the bearings of machine tools. — All bearings must be regularly oiled. Use good lubricating oil, — machine oil, not lard oil. If the oil does not sink into the oil holes, they should be cleared out. A few drops of oil in each bearing is enough, Fig. 7. Plane bearings, as the

FIG. 7. — OILING SPINDLE OF A MACHINE.

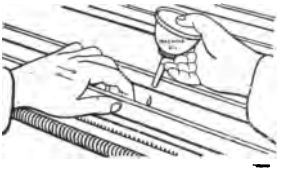


FIG. 8. — OILING WAYS OF A MACHINE.

ways of a lathe, should be wiped with waste before oiling and the oil distributed with the fingers, Fig. 8. Oil twice a day. All automatic oilers should be filled periodically.

28. Treatment when bearings rough up and machine stops from lack of oil or too close adjustment. First force in a liberal quantity of oil; if this does not release the bearing, force in naphtha or benzine and then more oil; if the latter is not effective, take the bearing apart and smooth the rough places on the journal by filing and those in the box by scraping or filing. Then wipe clean, oil freely, and put bearing together and adjust to run loosely for a while.

29. To prevent rusting or corroding of machine or finished work, coat with vaseline or thin oil. When tools or machines are not in use, coat with a thick oil. The rust should be removed from a surface before it is oiled or rusting will continue.

30. Putting on and pushing off belts. — An ordinary overhead belt is thrown off by pressing with a pole against the edge of the belt at the receiving side of driving pulley. This belt may be replaced by arranging belt upon driven pulley and then pushing belt upon driving pulley by starting it at the

Digitized by Google

receiving side with a pole that has a bent piece of iron fastened to its end. For large belts, a ladder must be used and the belt pushed on by hand. A stout cord may be used to pull on large belts.

31. To change to higher speed. — Start lathe, push belt off step of head cone with right hand. Press lightly against inside of down-running belt with either hand to take up slack and with other hand push the up-running side of belt to the desired step on counter cone, then with left hand press belt on to head cone.

32. To change to lower speed. — If belt is on largest step of counter cone, with left hand pull it on to desired step, then with same hand press belt on to corresponding step of head cone. If, however, belt is not on largest step, first push it off step of head cone with right hand.

Attention. — Keep fingers straight and stiff and do not try to hold belt when pushing on or off.

Warning. — To prevent accidents, do not wear loose sleeves or be careless in your actions near running belts, gears, milling cutters, etc.

Digitized by Google

CHAPTER II.

TRUING AND ALINING CENTERS. CENTER HOLES IN LATHE WORK. HAND AND MACHINE METHODS OF CENTERING.

TRUING AND ALINING CENTERS.

33. Center gage *A*, Fig. 9, is used for defining angles of 60°. The large notch is used for testing lathe centers, as *B*; notches *C* and *D* for testing and setting outside threading

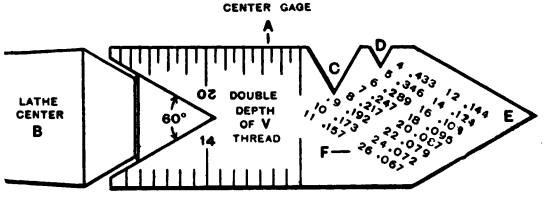


FIG. 9. — TESTING ANGLE OF LATHE CENTER.

tools, and *E* for inside threading tools. At *F* is a table of double depths of Sharp *V* threads for determining diameter of tap drills by subtracting number in thousandths opposite pitch from diameter of tap.

34. Requirements for successful use of engine lathes. — Engine lathes should be accurate enough to turn, bore and face straight and true, and be equipped with an accurate lead screw for screw cutting.

To produce accurate work requires that both live and dead centers should be true and in accurate alinement. Furthermore, lathes wear, causing loss of alinement and looseness of working parts that must be detected and corrected.

35. Lathe centers. — The dead center is hardened and tempered. The live center may or may not be hardened.

Digitized by Google

To test truth of live center of any lathe. — Move footstock until dead center is close to live center, run lathe at highest speed, look for error. Move lathe tool close to revolving live center or use test indicator.

36. To true engine-lathe centers. Figs. 10 and 11.

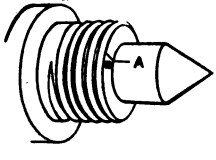


FIG. 10. — LOCATING CENTER IN HEADSTOCK SPINDLE.

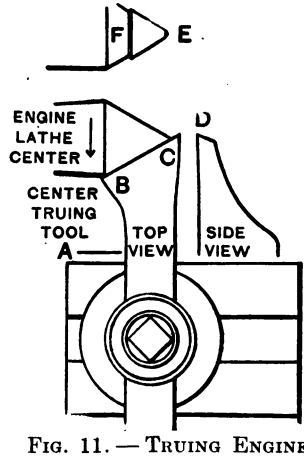


FIG. 11. — TRUING ENGINE-LATHE CENTER.

SCHEDULE OF OPERATIONS.

Live Center.

1. Remove center.
2. Clean hole and center with waste.
3. Insert center with lines coincident *A*, *B*, Fig. 10.
4. Drive center lightly with lead hammer.
5. Start lathe; if still out of true, use center-truing tool *A*, Fig. 11. *BC* shows cutting edge, *D* the clearance.
6. Fasten tool lightly at height of center.
7. Adjust edge to fit center.
8. Clamp tool tightly.
9. Run lathe at moderate speed.
10. Operate both long. and cross feeds slowly by hand.
11. Test with center gage. Readjust tool, if necessary, until center fits gage.
12. Turn point *E* at a more obtuse angle. Portion *F* may be cut away in advance to facilitate truing center.

Attention. — Centers may be trued by means of a compound rest. In an emergency a right side tool may be used back of the center and the lathe run backward.

Digitized by Google

SCHEDULE OF OPERATIONS. *Concluded.*

Dead Center.

1. Remove center.
2. Anneal.
3. Insert in live spindle.
4. True similarly to live center.
5. Run lathe at high speed.
6. File center with 8″ mill file.
7. Polish slightly with fine emery cloth held under file.
8. Reharden.
9. Temper to light straw color.

Attention. — Both centers may be hardened and kept true by grinding.

37. To grind hardened and soft centers. Fig. 12.

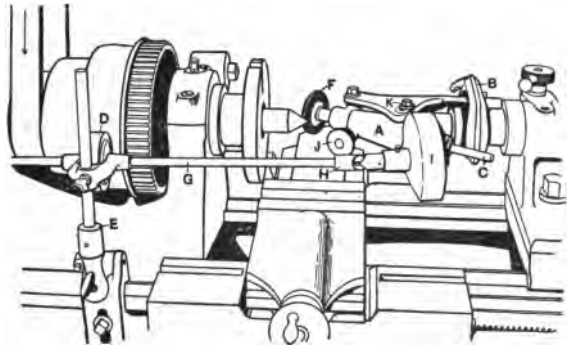


FIG. 12. — TRUING CENTERS WITH CENTER GRINDER.

SCHEDULE OF PARTS.

A — Frame.
B — Clamp fastening *A* to footstock spindle.
C — Emery-wheel spindle set at angle of 60° with center.
D — Friction roll.
E — Bracket; clamps support for *D* to bed.
F — Emery wheel.
G — Shaft driving *F*.
H — Universal joint.
I — Bevel gears (encased).
J — Knob to move emery wheel back and forth along center.

Attention. — The depth of cut is regulated by footstock spindle.

Digitized by Google

38. To true speed-lathe centers, Fig. 13, set rest *A* close to center *B* and adjust till graver *C* is at height of center. Turn with graver and test with gage. The dead center is first

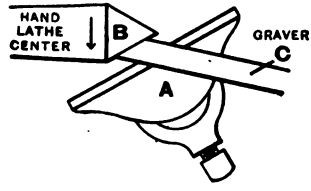


FIG. 13. — TRUING SPEED-LATHE CENTER.

annealed, inserted in live spindle and trued; then filed, polished, hardened, and tempered.

39. To set dead center in alinement to turn straight.

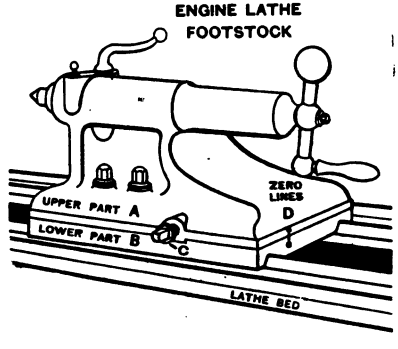


FIG. 14. — SETTING DEAD CENTER IN ALINEMENT. APPROXIMATE METHODS.

SCHEDULE OF OPERATIONS.

40.	*Two Approximate Methods.*
I. Unclamp footstock, Fig. 14, and move upper part *A* upon *B* by screws *C* and *C'* (*C'* not shown) until zero lines at *D* coincide.	II. Move footstock until dead center is close to live center, adjust screws *C* and *C'*, and aline centers by sight.

Digitized by Google

41. *Accurate Method.*

1. Set dead center by approximate methods either No. I or No. II.
2. Mount shaft *A* on centers, Fig. 15.
3. Take light cut as shown dotted at *B*.
4. Take out work.
5. Run carriage back near dead center.
6. Remount work.
7. Take short cut as at *C*.
8. Caliper at *B* and *C* with micrometer.
9. If not alike, adjust footstock and repeat operations.

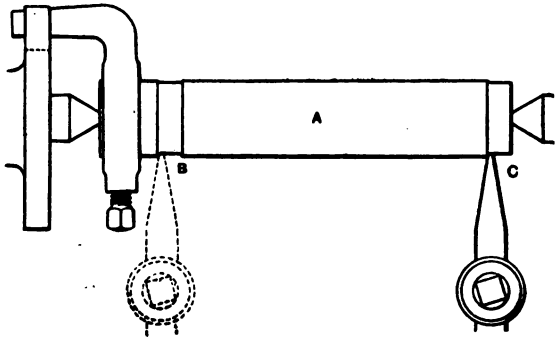


FIG. 15. — SETTING DEAD CENTER IN ALINEMENT. ACCURATE METHOD.

Attention. — Limit of error permissible: *B* may be a fraction of a thousandth of an inch larger than *C* but never smaller.

This insures the lathe turning slightly large, which is permissible for ordinary work.

Note. — A test indicator and parallel mandrel, preferably of the length of the work to be turned, may also be used to set a lathe to turn straight.

CENTER HOLES IN LATHE WORK.

42. Center holes are made in the ends of stock to fit lathe centers by locating and drilling small holes, then countersinking with a 60° countersink, *A* and *B*, Fig. 16.

The countersink should be large enough to provide ample

Digitized by Google

bearing to prevent excessive wear, and should be in proportion to the diameter of work.

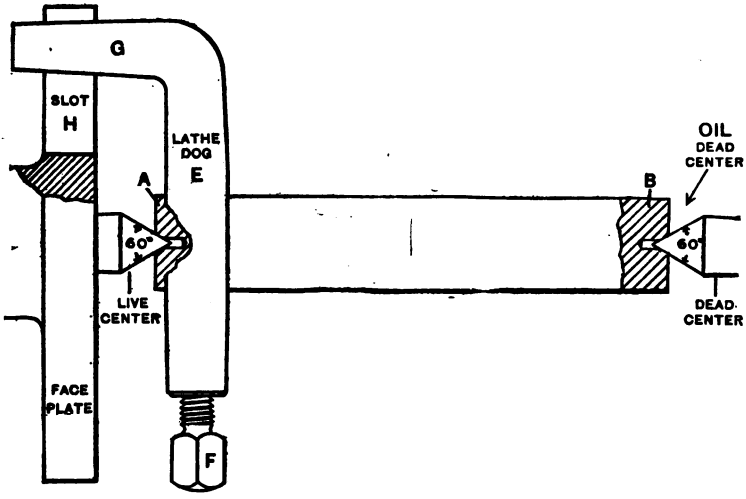


FIG. 16. — STOCK CORRECTLY CENTERED AND PROPERLY MOUNTED ON LATHE CENTERS.

The drilled hole must be deeper than countersink to provide a reservoir for oil and to prevent lathe centers from bottoming, as at *C*, Fig. 17, as this would injure centers and

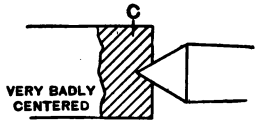


FIG. 17. — INCORRECT CENTERING.

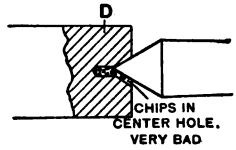


FIG. 18. — STOCK CARELESSLY MOUNTED ON CENTERS.

cause work to run out of true. Center holes must be wiped clean before mounting on centers. Chips in center holes *D*, Fig. 18, will spoil both work and center.

43. To drive the work, dog *E*, Fig. 16, is fastened to work by screw *F*, and tail *G* must be loose in face-plate slot *H*.

Digitized by Google

44. Table of center-hole dimensions. Fig. 19. — These center-hole dimensions provide good bearings for ordinary lathe work to resist tool pressure.

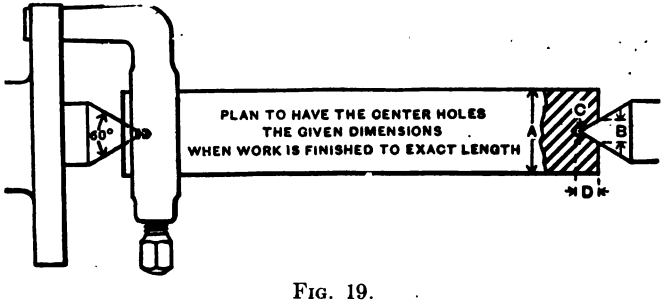


FIG. 19.

DIAMETER OF SHAFT.	DIAMETER OF COUNTER-SINKS.	DRILL SIZE.		DEPTH OF HOLE = $B + \frac{1}{16}''$.
		NEAREST 64TH.	GAGE NO.*	
A.	B.	C.		D.
$\frac{1}{8}$–$\frac{3}{16}$	$\frac{5}{64}$	$\frac{3}{64}$	56	$\frac{9}{64}$
$\frac{1}{4}$–$\frac{7}{16}$	$\frac{3}{32}$	$\frac{1}{16}$	52	$\frac{5}{32}$
$\frac{1}{2}$–$1\frac{1}{4}$	$\frac{1}{8}$	$\frac{1}{16}$	52	$\frac{5}{32}$
$\frac{3}{4}$–$1\frac{5}{8}$	$\frac{5}{32}$	$\frac{3}{32}$	42	$\frac{7}{32}$
1–$1\frac{3}{16}$	$\frac{3}{16}$	$\frac{3}{32}$	42	$\frac{1}{4}$
$1\frac{1}{4}$–$1\frac{7}{16}$	$\frac{7}{32}$	$\frac{3}{32}$	42	$\frac{9}{32}$
$1\frac{1}{2}$–$1\frac{11}{16}$	$\frac{1}{4}$	$\frac{1}{8}$	31	$\frac{5}{16}$
$1\frac{3}{4}$–$1\frac{15}{16}$	$\frac{9}{32}$	$\frac{1}{8}$	31	$\frac{11}{32}$
2 –$2\frac{7}{16}$	$\frac{5}{16}$	$\frac{1}{8}$	31	$\frac{3}{8}$
$2\frac{1}{2}$–$2\frac{15}{16}$	$\frac{11}{32}$	$\frac{1}{8}$	31	$\frac{13}{32}$
3 –$3\frac{7}{16}$	$\frac{3}{8}$	$\frac{5}{32}$	22	$\frac{7}{16}$
$3\frac{1}{2}$–$3\frac{15}{16}$	$\frac{7}{16}$	$\frac{5}{32}$	22	$\frac{1}{2}$
4 –$4\frac{7}{16}$	$\frac{1}{2}$	$\frac{5}{32}$	22	$\frac{9}{16}$
$4\frac{1}{2}$–$4\frac{15}{16}$	$\frac{9}{16}$	$\frac{5}{32}$	22	$\frac{5}{8}$
5 –$5\frac{7}{16}$	$\frac{5}{8}$	$\frac{5}{32}$	22	$\frac{11}{16}$
$5\frac{1}{2}$–6	$\frac{11}{16}$	$\frac{3}{16}$	13	$\frac{3}{4}$

Attention. — If drill and countersink are within one size they will answer. Over 6″, countersink may be made $\frac{1}{4}$ diameter of work and drill $\frac{1}{8}$ diameter of countersink.

* Twist drill and steel wire gage.

Digitized by Google

45. Countersinks or center reamers, Figs. 20 and 21, are made to an angle of 60°. They may be made any desired diameter and with straight shanks, as shown, to be held in a chuck, or with taper shanks to be inserted in taper collet or machine spindle. They may have several cutting lips as in Fig. 20, or but a single cutting lip as in Fig. 21.

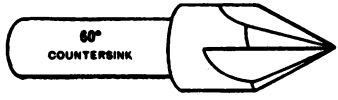


FIG. 20.

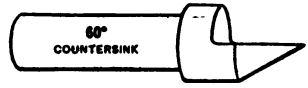


FIG. 21.

COUNTERSINKS FOR LATHE WORK.

FIG. 22. — COMBINED DRILL AND COUNTERSINK.

Combination center drill and countersink, Fig. 22, produces a countersink central with drilled hole. At end *A* drill and countersink are one piece, while at end *B* a hole is made in countersink and a center drill inserted and held by a set screw.

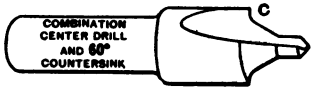


FIG. 23. — COMBINED DRILL, COUNTERSINK, AND COUNTERBORE.

Fig. 23 shows a combination center drill and countersink of the solid type as at end *A*, Fig. 22, and in addition a counterboring lip as at *C* for rounding the corners of the countersink. Combination countersinks of this typ e are used for centering mandrels, milling machine arbors and any other work that is mounted and remounted on centers frequently. Their use also facilitates the squaring of work.

Digitized by Google

HAND AND MACHINE METHODS OF CENTERING.

46. Hand method of finding the center, drilling and countersinking. Figs. 24, 25, 26, 27, 28 and 29.

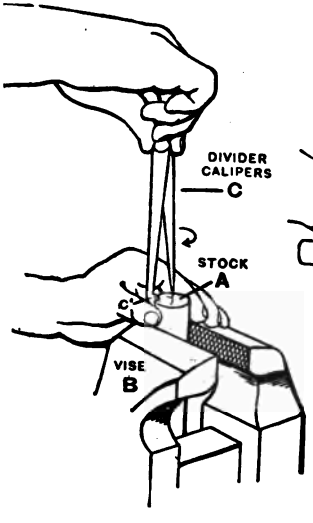


FIG. 24. — LOCATING CENTER OF STOCK WITH DIVIDER-CALIPERS.

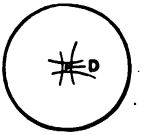


FIG. 25. — DIAGRAM OF APPROXIMATE CENTER.

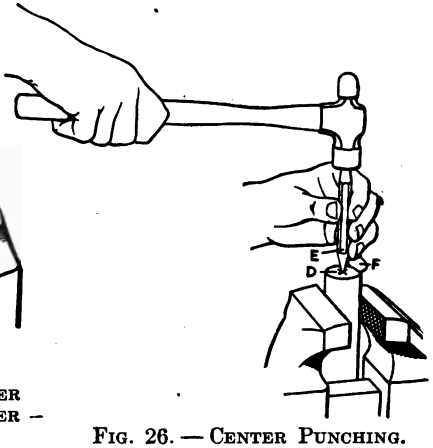


FIG. 26. — CENTER PUNCHING.

SCHEDULE OF OPERATIONS.

I. Grip stock *A*, Fig. 24, in vise *B*. Smooth ends with file and rub chalk on ends.

II. Describe arcs with divider calipers *C* (shoulder *C'* placed on edge of stock) from four points with radius equal to about one-half diameter, as at Fig. 25.

III. Locate center *D*, Figs. 25 and 26, by eye with center punch *E*, Fig. 26. Steady with finger *F* and strike with hammer as shown.

IV. Mount on bench or lathe centers, revolve and test its truth with chalk near each end. If too much out of true, set over center punch mark as at *G*, Fig. 27. Repeat if necessary.

Digitized by Google

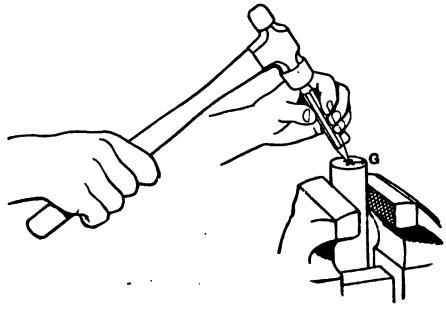


FIG. 27. — SETTING OVER PUNCH MARK.

SCHEDULE OF OPERATIONS. *Concluded.*

V. Enlarge center punch marks with heavier blows before drilling.

VI. Place drill *A* in drill chuck *B*, Fig. 28, and end of stock *C* on dead center. Support with hand, start lathe, and drill. Use speed lathe, 3d or 4th speed.

Attention. — Withdraw drill occasionally to let out chips, and oil into hole. Drill cast iron or brass dry; on steel or wrought iron, use oil.

VII. Place countersink *D* in drill chuck *E*, Fig. 29, hold stock *F* as before, start lathe and countersink to desired depth. 1st or 2d speed.

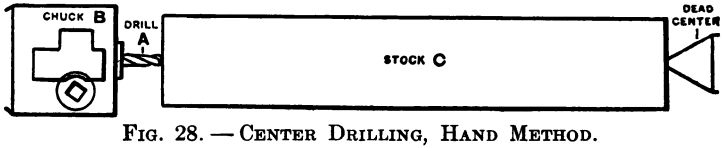


FIG. 28. — CENTER DRILLING, HAND METHOD.

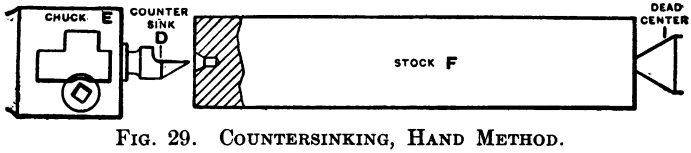


FIG. 29. COUNTERSINKING, HAND METHOD.

47. To remove or anneal broken center drill. — If the broken piece of drill does not drop out of hole with the aid of a scratch awl, strike stock with hammer; if it still remains, the work must be annealed, and the hole redrilled.

Digitized by Google

48. Machine method of finding the center, drilling and countersinking. In Fig. 30 is a centering machine used to drill and countersink center holes accurately in shafts of any length.

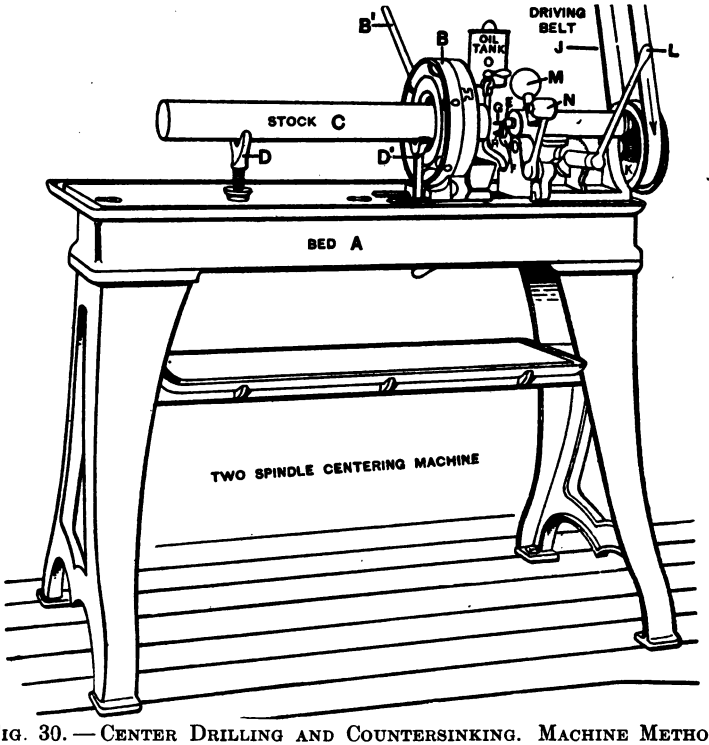


FIG. 30. — CENTER DRILLING AND COUNTERSINKING. MACHINE METHOD.

CENTERING MACHINE. SCHEDULE OF PARTS AND OPERATIONS.

A — Bed bolted to floor.	*K* — Gear mechanism which drives *E* and *F* at different speeds.
B and *B'* — Universal chuck and operating lever.	*L* — Handle operating both spindles.
C — Shows manner of holding stock to be centered.	*M*—Ball lever for tipping head to bring either spindle into alinement with stock.
D and *D'* — Supports.	*N* — Stock or work stop for uniform drilling.
E — Drill spindle.	*O* — Oil tank.
F — Countersink spindle.	
G — Center drill.	
H — Countersink.	
J — Driving belt.	

Digitized by Google

Precaution. — The end of stock should be approximately flat and smooth, or the drill will be liable to break.

49. Straightening shafting, rods, and bolts. — As soon as stock is centered, it is mounted on centers and revolved by hand and its eccentricity tested with chalk. If it is short or rigid, it may have to be straightened with a hammer on the anvil; but if slender, it may be straightened on lathe centers or with straightening press. See *Elements of Machine Work.*

Digitized by Google

CHAPTER III.

CUTTING TOOLS. LATHE TOOLS FOR CAST IRON. GRINDING LATHE TOOLS.

CUTTING TOOLS.

50. All cutting tools may be considered primarily as wedges driven into the material to separate it. A thin-edge tool cuts more easily, because it generates less friction, distorts the chips less and gives a greater freedom to their removal. The edge must be thick enough to carry a heavy cut at a suitable speed, and have a point of sufficient width to stand the heat generated by friction. Excessive heat will soon destroy the point of the tool. This limits the cutting speeds. A portion of the heat is conducted away through the work to the air and from point of tool to body and to air by direct radiation, and on tenacious metal by the lubricant. See Lubricants for Cutting Tools, § 100.

51. Rake, clearance, and cutting angle defined. — *Rake* is applied to angle of upper surface and *clearance* to angle of lower surface. The angle included between these surfaces is the *cutting* angle, or angle of keenness.

52. Front rake, end clearance, and cutting angle. — Fig. 31 is a side view of a square-nose tool partially cut by a plane giving a section CAE. Through point A lines AB and AD are drawn parallel and perpendicular to the base line FG. The front-rake angle is BAC, positive when below and negative when above AB. The end-clearance angle is EAD. The cutting angle is CAE.

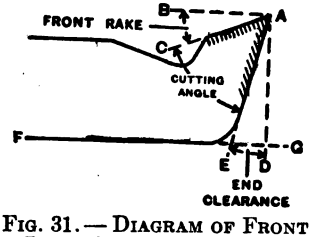


FIG. 31. — DIAGRAM OF FRONT RAKE, CLEARANCE, AND CUTTING ANGLE.

Digitized by Google

53. Side rake, side clearance, and cutting angle. — Fig. 32 is an end view of a right-side tool partially cut by a plane, giving section *K HM*. Through point *H* lines *HJ* and *HL* are drawn parallel and perpendicular to the base of tool *NO*. The side-rake angle is *JHK*. The side-clearance angle *MHL*. The cutting angle is *KHM*.

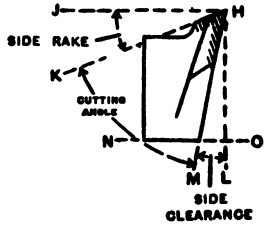


FIG. 32.—DIAGRAM OF SIDE RAKE, SIDE CLEARANCE, AND CUTTING ANGLE.

54. The keenness of a cutting edge is increased or decreased by increasing or decreasing the rake angle. Cast iron may be cut successfully with a cutting angle of from 60° to 75°; steel and wrought iron, with a cutting angle of from 40° to 50°. See Brass Turning, § 357.

55. Front and side rake combined. — For clearness the front rake and side rake are shown on separate tools, but some turning tools will cut more effectively if the top face is given a combination front and side rake in varying degrees to suit the nature of the work, as the diamond-point tool, Fig. 45.

56. "Rough turn" and "rough square." — Terms used to name the operations of removing the surplus material from a piece of metal by one or more roughing cuts preparatory to finishing.

57. "Finish turn" and "finish square." — Terms used to name the final finishing cuts which reduce any piece of metal to required size.

LATHE TOOLS FOR CAST IRON.

58. Lathe tools are made of carbon steel and high-speed steel. See Holders and Cutters, §§ 192, 194 and *Elements of Machine Work*.

59. A chart of forged lathe tools is shown in Fig. 33. For 10″ to 12″ lathes, they are made $\frac{3}{8}$″ × $\frac{3}{4}$″ in section and 7″ in length; for 14″ to 16″ lathes, $\frac{1}{2}$″ × 1″ section, 9″ in length. Other sizes in proportion. After being forged it is best to file them to proper shape before they are hardened.

Digitized by Google

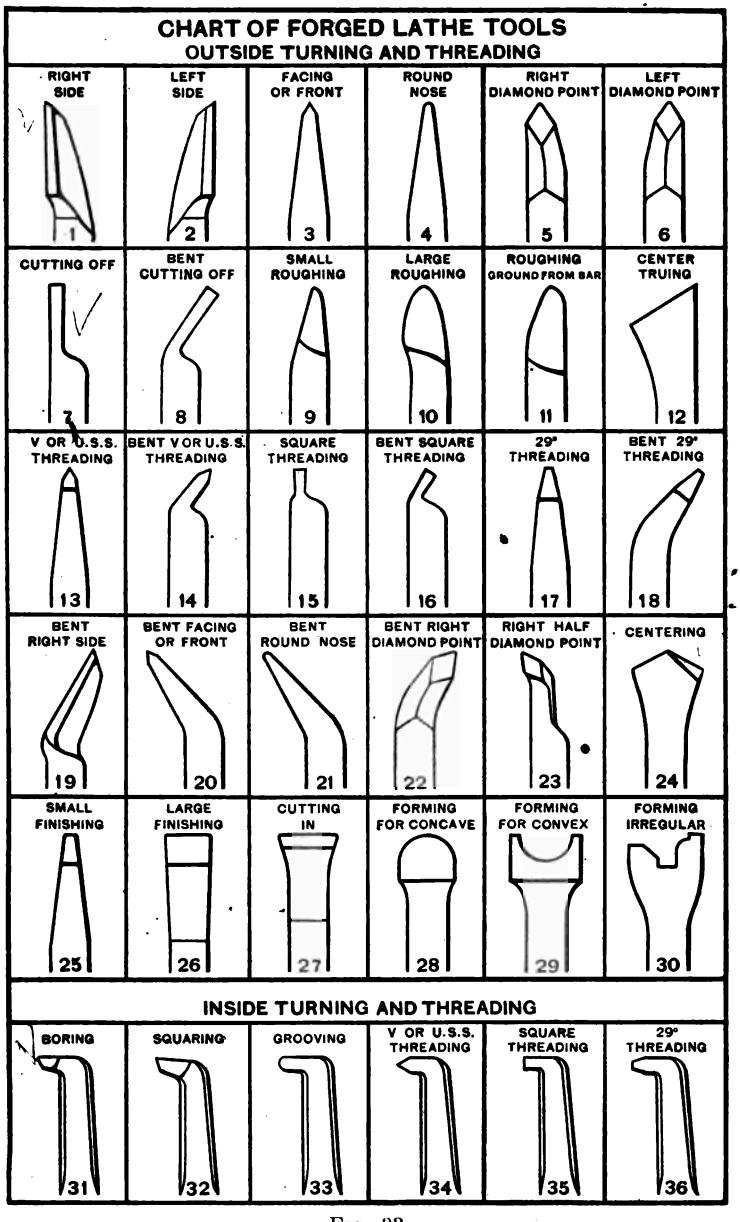


FIG. 33.

Digitized by Google

60. Right and left tools. — A *right* tool cuts from right to left and a *left* tool from left to right. Tools are understood to be *right* unless otherwise designated.

61. The angles of lathe cutting tools. — In order to select the proper tool and prepare the correct cutting angle, the student should consider the kind and, if possible, the hardness of the metal and whether for taking a roughing or finishing cut. If the metal is very hard, the tool must be ground to a less acute cutting angle, the cutting speed reduced, or both.

62. Height of tool and tool block. — Various devices are used to regulate the height of the point of the tool. On

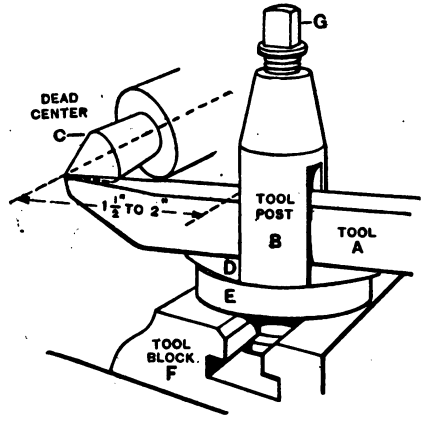


FIG. 34. — SETTING TOOL HEIGHT OF CENTERS — PLAIN REST.

small lathes the rise and fall rest operated by an elevating screw is perhaps the most common. Fig. 34 shows a plain rest. The point of tool *A* is adjusted in tool-post *B* to height of dead center *C* by a tilting action of circular wedge *D* in concave washer *E* and the shank clamped by screw *G*.

Digitized by Google

63. Round-nose tool. — Fig. 35 shows a small round-nose tool used for roughing and finishing cast iron or brass. Face *A* has no rake, but the sides have 10° clearance. The point is about $\frac{1}{8}$″ thick. *B* is the cutting edge and *CBD* the clearance angle. Too little clearance will cause the tool to ride on the work and too much will weaken the cutting edge. When dull, grind end *B* and a little on top *A*.

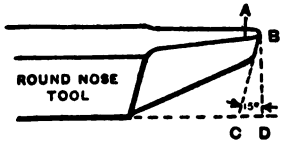


FIG. 35.— ROUND-NOSE TOOL FOR CAST IRON.

64. To square with round-nose tool. — The scale or skin on cast iron is very hard, and the round-nose tool is used to rough square ends and remove surplus stock, as in Fig. 36. Cast iron is machined dry. See Lubricants for Cutting Tools, § 100.

The work is mounted on centers and the lathe run at proper speed; arrow 1 shows direction of rotation and arrow 2 direction of cut. The long. feed handle is held firmly with one hand, while the tool is fed with the other operating the cross-feed handle.

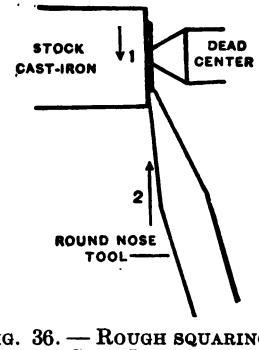


FIG. 36. — ROUGH SQUARING CAST IRON.

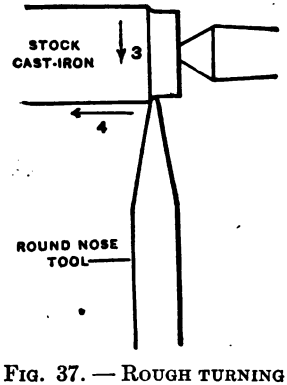


FIG. 37. — ROUGH TURNING CAST IRON.

65. To turn with round-nose tool. — For light rough turning on small diameters and for finish turning with fine feed, a round-nose tool may be used to advantage on cast iron, as in

Digitized by Google

Fig. 37. Arrow 3 shows direction of rotation of work and arrow 4 direction of cut. It is sometimes necessary to slant the tool to the left to turn close to a shoulder or dog, but the tool must be clamped extra firm or it may draw into the work and turn the diameter too small.

In Fig. 38 is shown a large round-nose tool for turning or facing large work. It is ground to shape from the bar and given side rake as at *A*, to give freedom to removal of chips.

Fig. 38. — Tool for Heavy Cuts, Rough squaring or Turning Cast Iron.

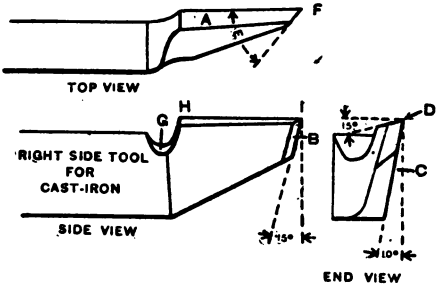


Fig. 39. — Side Tool for Squaring Cast Iron.

66. Side tool. — For squaring or facing the ends of shafts, shoulders, etc., a right-side tool, Fig. 39, is used. The tool has end clearance *B*, 15°; side clearance *C*, 10°; and side rake *D*, 15°. The angle *E* for point *F* is 60°. It is forged hollow at *G*, to facilitate grinding. Grinding is done on top *A* and end *B* with a little on side *C*. Cutting edge *HI* should be kept horizontal.

Fig. 40 shows a right-side tool suitable for heavy work.

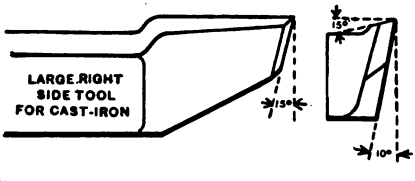


Fig. 40. — Side Tool for Heavy Cuts, Squaring Cast Iron.

Digitized by Google

67. To square an end with side tool. — A side tool is set at the height of the center, as in Fig. 41. On diameters not larger than $\frac{3}{8}$", edge AB is set at right angles to the axis of work, so as to square the whole end at one cut. For large diameters the point should " drag " a little, as at A, Fig. 42,

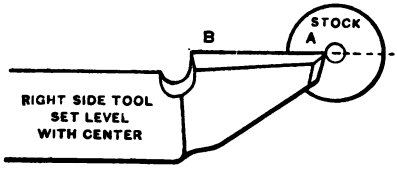


FIG. 41. — SIDE TOOL SET HEIGHT OF CENTERS.

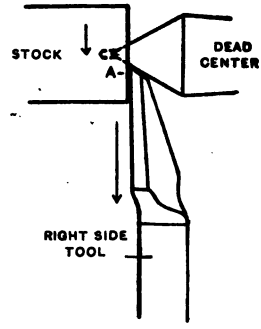


FIG. 42. — FINISH SQUARING CAST IRON.

for both roughing and finishing cuts. The point A being slightly the deepest, the tool when carried from center to circumference will produce a smooth surface, provided the tool is properly hardened, tempered, and ground, and the speed and feed are correct. For some purposes, especially in squaring compositions of brass, a side tool is fed inward.

68. To remove burr around countersink. — To remove the burr that remains around the countersink after taking the finishing cut, feed the point of tool up to surface of work and close to dead center; then unclamp binder and relieve dead center slightly with right hand and at same moment slightly feed tool inward with left hand, which will remove burr; then simultaneously feed tool outward and dead center back in place.

69. Grooved dead center for squaring. — The extra operation of removing burr around countersink when squaring may be avoided and time saved by using a grooved dead

Digitized by Google

center *A*, Fig. 43. As point of tool *B* may be started or terminated in groove, no burr remains.

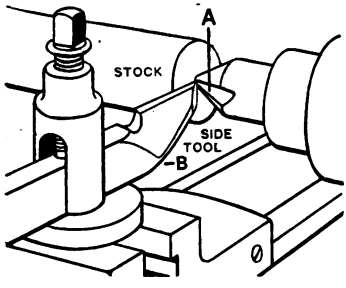


FIG. 43. — GROOVED DEAD CENTER FOR SQUARING.

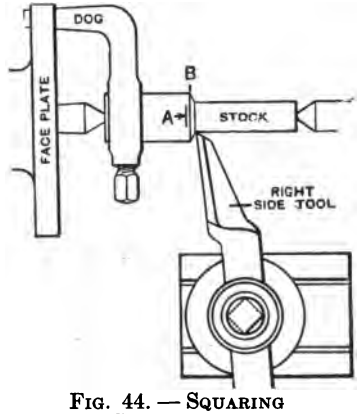


FIG. 44. — SQUARING SHOULDER.

70. To square a shoulder with side tool. — To turn a portion of a piece of stock and square the shoulder, as in Fig. 44, it is marked as at *A*, then the cut taken to *B*, and the shoulder squared to mark *A*. The side tool is fed inward to touch the stock. A moderately fast speed is used and the long. feed is fed slowly with one hand, while the cross feed is held firmly with the other; when the cut is carried far enough, the long. feed is held firmly and the cross feed fed outward.

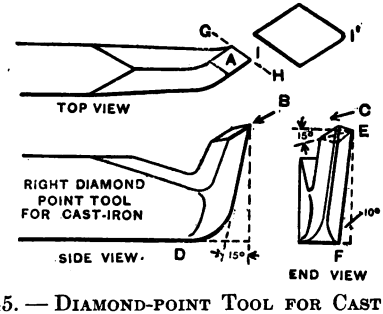


FIG. 45. — DIAMOND-POINT TOOL FOR CAST IRON.

71. Right diamond-point tool. — Fig. 45 shows a right diamond-point tool. *A* is the top face, which is given a com-

Digitized by Google

bination front and side rake, as indicated by arrows *B* and *C*. Side clearance *EF* is 10°, but for a very coarse feed should be more.

Cast iron of small diameter may be turned by tools without rake, but for large diameters and heavy cuts a combination side and front rake of about 15° is effective. The cutting is done by edge *GH* and point *I*, which should be rounded, as shown enlarged at *I'*, to strengthen it and produce a smoother cut. The tool is ground on the top face *A*, and if necessary, a little on the side faces.

72. Height of lathe turning tools in relation to axis of work.— The point of taper turning and threading tools must be set at

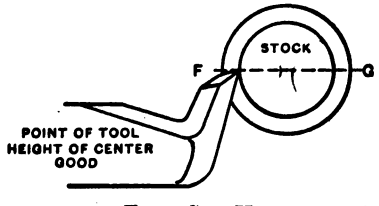


FIG. 46. — DIAMOND-POINT TOOL SET HEIGHT OF CENTERS. GOOD.

height of center, as at *FG*, Fig. 46. The student may apply this rule at all times and obtain good results.

73. Evil effects of setting a tool too low or too high. — A tool point set below the center *FG*, as exaggerated in Fig. 47, increases the clearance and decreases front rake, will not cut

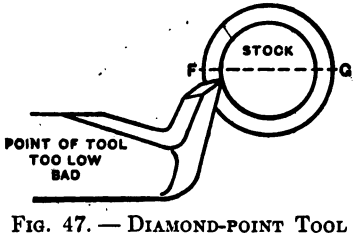


FIG. 47. — DIAMOND-POINT TOOL SET BELOW CENTERS. BAD.

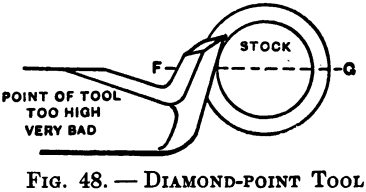


FIG. 48. — DIAMOND-POINT TOOL SET ABOVE CENTER. BAD.

properly, and will dull quickly. A tool point too high above center *FG*, exaggerated in Fig. 48, reduces the clearance, will ride on the work and soon destroy itself by friction.

Digitized by Google

74. Theoretical height of turning tools for straight work. — In Fig. 49 is shown the theoretically correct height to set the point of a tool, which increases its keenness and gives the greatest support to its cutting edge. This height is at the tangent point A of line BC, and is located by drawing line DE through center of work at 90° to line BC. As this gives no clearance, the tool in practice is set slightly below this point.

After a little training, one is able to set the tool point at the most suitable height in relation to center FG for any diameter of work.

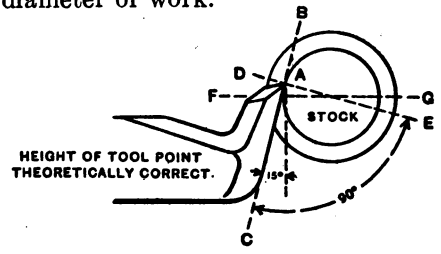


FIG. 49. — DIAMOND-POINT TOOL SET THEORETICAL HEIGHT. GOOD.

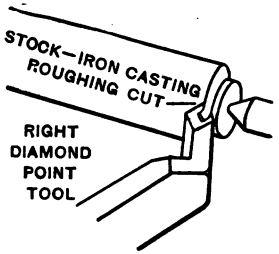


FIG. 50. — ROUGH TURNING CAST IRON, COARSE FEED.

75. Rough turning cast iron. — Fig. 50 shows a diamond-point tool taking a roughing cut by power long. feed on a cast-iron piece mounted on centers in an engine lathe. The chips from cast iron break off in small fragments.

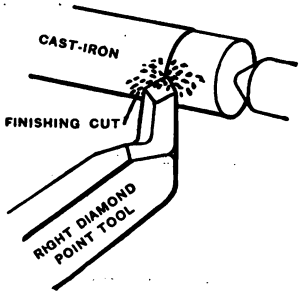


FIG. 51. — FINISH TURNING CAST IRON, FINE FEED.

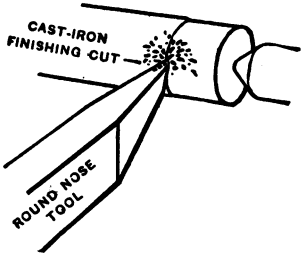


FIG. 52. — FINISH TURNING CAST IRON, FINE FEED.

76. To finish turn cast iron. — Fig. 51 shows diamond-point and Fig. 52 round-nose tool taking finishing cuts ($\frac{1}{64}$″)

Digitized by Google

with fine feed. Fig. 53 shows small square-nose tool and Fig. 54, large square-nose tool taking finishing cuts (.010″) with coarse feeds.

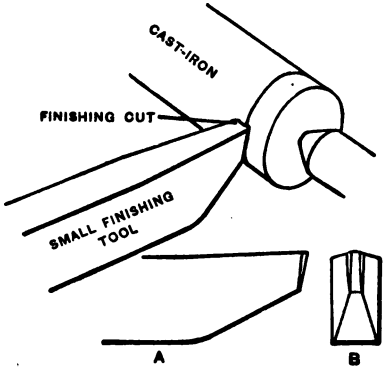


FIG. 53.— FINISH TURNING CAST IRON, MEDIUM FEED.

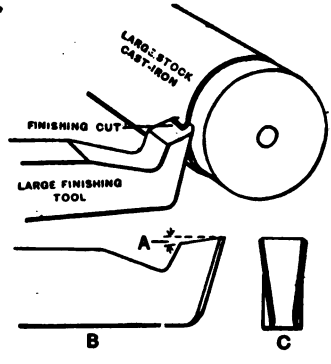


FIG. 54.— FINISH TURNING CAST IRON, COARSE FEED.

GRINDING LATHE TOOLS.

77. To grind round-nose tool. — Hold at a suitable inclination to give clearance, and first grind the front round by

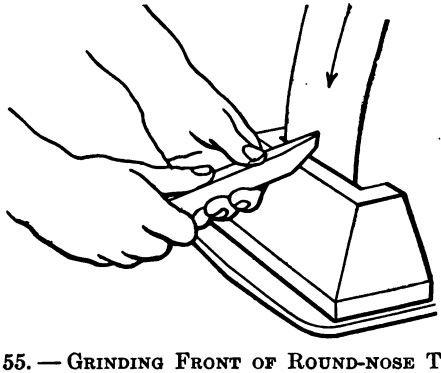


FIG. 55. — GRINDING FRONT OF ROUND-NOSE TOOL.

sweeping shank of tool in an arc of circle, rotating on heel, Fig. 55. Then grind top as in Figs. 56 or 57.

Attention.—For Grinding Wheels see §§ 384–397

Digitized by Google

For Wet Tool Grinders and Grindstones, see *Elements of Machine Work.*

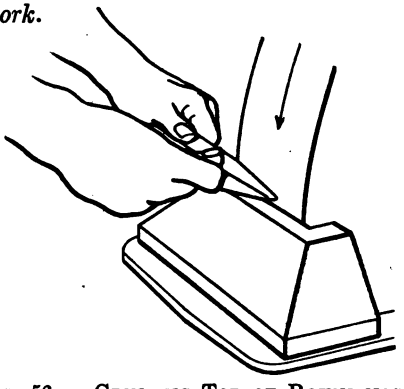


FIG. 56. — GRINDING TOP OF ROUND-NOSE TOOL.

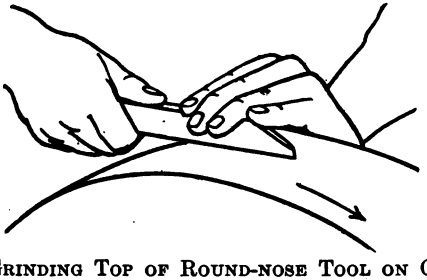


FIG. 57. — GRINDING TOP OF ROUND-NOSE TOOL ON GRINDSTONE.

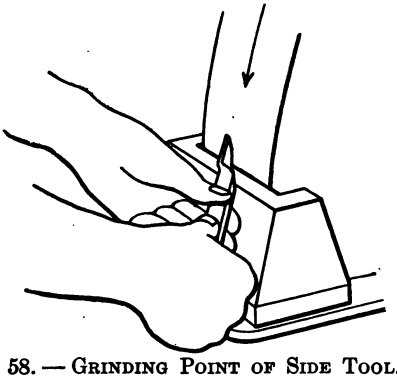


FIG. 58. — GRINDING POINT OF SIDE TOOL.

78. To grind right or left side tool. — First hold tool to give clearance and angle of point, and grind point, Fig. 58.

Digitized by Google

Second, hold blade at inclination to give necessary side rake and grind top face, Fig. 59. Apply pressure and steady with

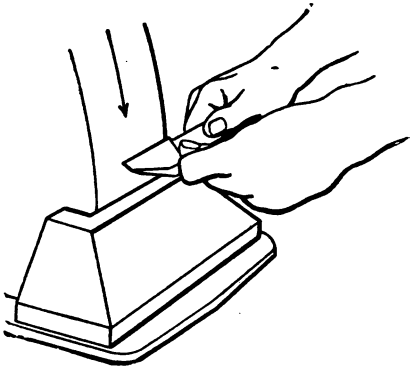


FIG. 59. — GRINDING TOP OF SIDE TOOL.

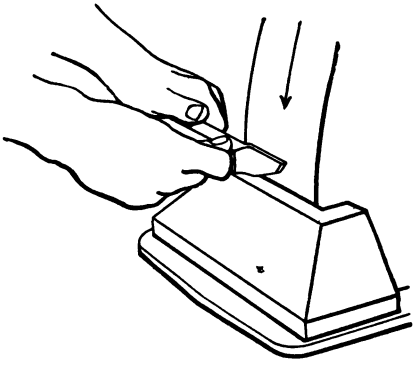


FIG. 60. — GRINDING SIDE FACE OF SIDE TOOL.

left hand. Third, grind side face, Fig. 60. Steady, and apply pressure with right hand.

Digitized by Google

79. To grind right or left diamond-point tool, Fig. 61. — Grind top face, applying pressure with left hand. Then hold

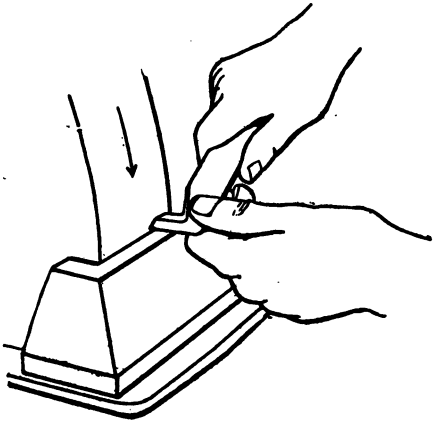


FIG. 61. — GRINDING TOP OF DIAMOND-POINT TOOL.

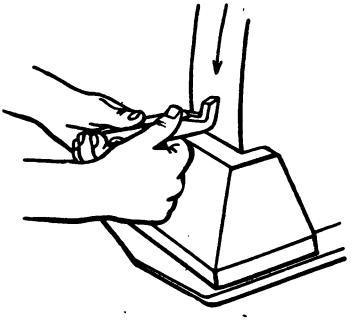


FIG. 62. — GRINDING SIDE FACE OF DIAMOND-POINT TOOL.

at angle to give clearance (same for all metals). Steady, and apply pressure with right hand; grind one side, Fig. 62; change hands and grind other side. Round point same as point of round-nose tool, Fig. 55.

80. To grind cutting-off or a Square-thread tool. — First file to shape and width, with proper clearance on sides and

Digitized by Google

end, harden and temper. Then grind on end and top face only. Grind end *A* as in Fig. 63. Steady, and apply

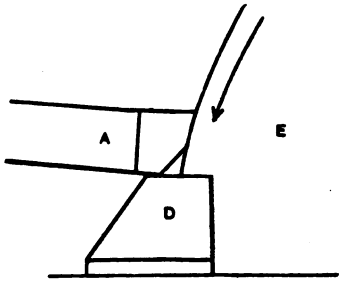


FIG. 63. — GRINDING END OF SQUARE-THREAD OR CUTTING-OFF TOOL.

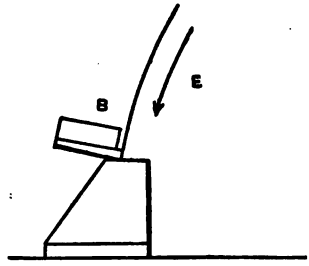


FIG. 64. — GRINDING TOP OF SQUARE-THREAD OR CUTTING-OFF TOOL.

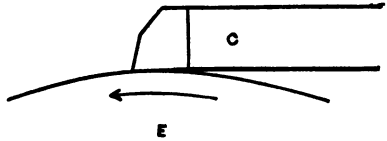


FIG. 65. — GRINDING TOP OF SQUARE-THREAD OR CUTTING-OFF TOOL ON GRINDSTONE.

pressure with left hand. Fig. 64, *B*, and Fig. 65, *C*, show two methods of grinding top face of any tool of this class.

81. To grind United States Standard or Sharp V-threading tool to fit a gage. — File tool to shape, harden and temper. Grind one bevel, as in Fig. 66. Clean gage and tool, and test angle.

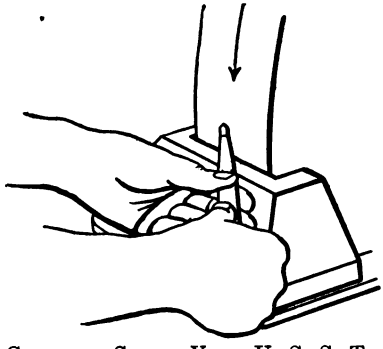


FIG. 66. — GRINDING SHARP V OR U. S. S. THREAD TOOL.

Digitized by Google

Grind second bevel; test with gage. If U. S. S. tool, grind point to fit proper notch in U. S. S. gage. Grind top face.

82. To grind a carbon or high-speed steel removable cutter, clamp cutter in tool holder, as in Fig. 67, and grind principally on end and a little on top for rake. For other purposes, grind cutters similarly to forged tools.

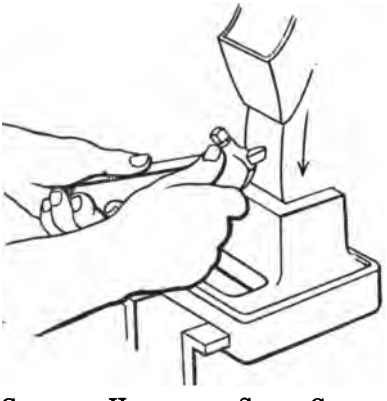


FIG. 67. — GRINDING HIGH-SPEED STEEL CUTTER ON A WET TOOL GRINDER.

FIG. 68. — GRINDING HAND GRAVER.

83. To grind graver. Fig. 68. — Hold tool at inclination to produce 60° bevel on end. Grind the sides a little.

Digitized by Google

84. To grind round-nose hand tools, apply them as described in § 77 and illustrated in Figs. 55, 56, and 57.

85. To grind boring tool *A*, Fig. 69, use corner of wheel to grind rake *B* and face of wheel to grind point and clearance.

86. Universal tool grinder. — Machine method of grinding lathe and planer tools, and duplicating angles.

Fig. 70 shows how to grind rake of a right diamond-point tool. Clamp diamond-point *A* in holder *B*. Set three graduated circles, *C*, *D*, and *E*, to readings obtained from chart

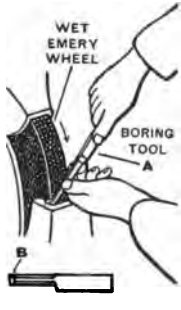


FIG. 69. — GRINDING BORING TOOL.

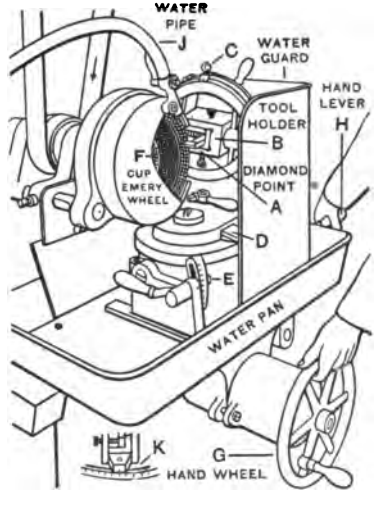


FIG. 70. — DUPLICATING ANGLES OF LATHE AND PLANER TOOLS.

(chart furnished with machine), or obtain setting by trial. Move tool to radial face of cup-shaped emery wheel *F* with hand wheel *G*. Carry tool back and forth across wheel with lever *H*. Water is supplied from pipe *J*.

A fourth graduated circle on rear of tool holder, as shown in detail at *K*, is used in grinding bent tools.

To grind side faces, revolve holder *B* and set dials to give proper angles. To reduce area to be ground, tools may be

Digitized by Google

forged in former blocks, or by hand to more clearance than desired. The method of grinding rake of a right side tool is shown in Fig. 71. When correct angles and settings of tools are obtained, they can be accurately duplicated.

Round-nose or circular forming tools are located centrally with a gage and ground by swinging on a vertical axis. The top is ground in the usual way.

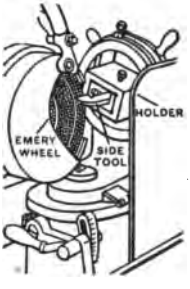


FIG. 71. — GRINDING SIDE TOOL.

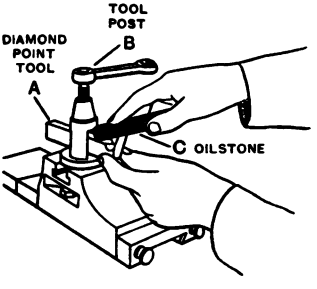


FIG. 72. — OILSTONING LATHE TOOLS.

87. To oilstone tools. — Use fine manufactured or hard Arkansas stone about $4'' \times \frac{3}{4} \times \frac{3}{4}''$. Clamp diamond-point tool *A* reversed in tool-post *B* and apply oilstone *C* as in Fig. 72, with long strokes. Also oilstone side faces. Use kerosene oil for manufactured, and lard oil for Arkansas stones.

Digitized by Google

CHAPTER IV.

SETTING AND USING OUTSIDE CALIPERS. CUTTING SPEEDS, CUT-METER, AND FEEDS. LUBRICANTS FOR CUTTING TOOLS. INSPECTING AND MEASURING STOCK. OILING AND CLEANING MACHINES.

SETTING AND USING OUTSIDE CALIPERS.

88. To set outside calipers *A*, Fig. 73, by rule *B* to length *CD*, adjust nut *E* until lower point coincides with middle of line *D*. The width of lines on steel rules is from .002″

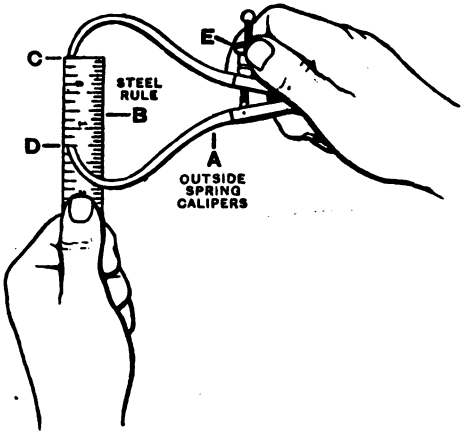


FIG. 73. — SETTING OUTSIDE CALIPERS.

to .004″. A student almost invariably sets outside calipers large and inside calipers small.

When extreme accuracy is required, such as turning work to be fitted, the calipers should be set by a standard plug gage or mandrel, or work of the desired diameter.

Digitized by Google

89. To measure diameter of lathe work with outside calipers. Fig. 74. — Set calipers *F* to size. Hold work *G* stationary and apply calipers at right angles to axis of work as *HI*, not as *JK* or *LM*. Turn work with tool *N* until calipers will pass over it with a light yet distinct touch, but

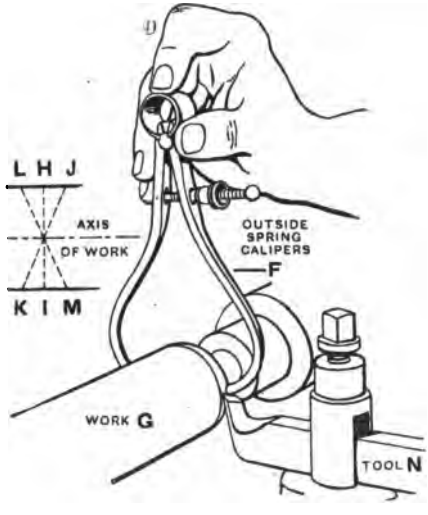


FIG. 74. — MEASURING WITH OUTSIDE CALIPERS.

not hard enough to spring calipers or to sustain their weight. Usually the calipers have to be passed over work a number of times to determine this touch.

90. To adjust the tool to turn work to a desired diameter. Fig. 74.— Move tool *N* inward at end of work to cut under scale, start lathe and feed tool to cut by hand, then throw in power long. feed and allow a travel of $\frac{1}{8}''$. Stop lathe and test diameter with calipers. If correct, continue turning; but if too large, start lathe, release power long. feed and run tool back to end and again slightly advance tool, throw in long. feed and so on until correct diameter is obtained.

Attention. — Hold long. feed firmly with one hand while releasing power feed with the other.

Digitized by Google

91. To learn to measure accurately with outside calipers. — Set calipers to rule, turn work until calipers will pass over it with a delicate touch, then test with micrometer calipers.

92. To transfer a setting from one pair of calipers to another. Fig. 75. — Set inside calipers *A* to size of hole, then set outside calipers *B* by them. Bring lower points of both calipers in contact and steady as at *C*, then adjust point *D*

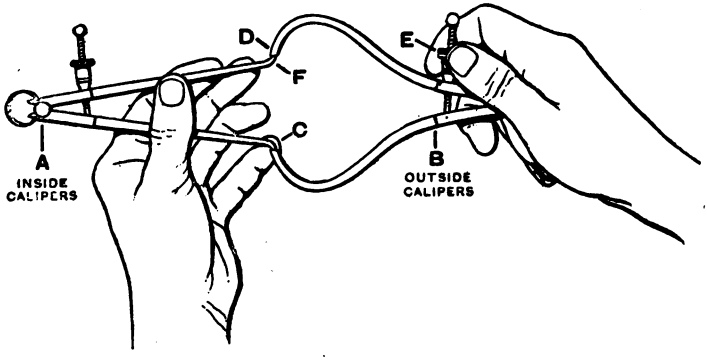


FIG. 75. — TRANSFERRING MEASUREMENT FROM INSIDE TO OUTSIDE CALIPERS.

by nut *E* until it touches point *F*. To transfer setting of outside to inside calipers, reverse calipers in hands and adjust as in Fig. 75.

CUTTING SPEEDS, CUT-METER, AND FEEDS.

93. In turning, three things must be considered:

First, the cutting speed in feet per minute, which is controlled by the diameter of work and speed obtained from the table; it is calculated or may be directly measured by a cut-meter.

Second, the depth of cut, one-half the amount that the diameter is reduced.

Third, the feed or amount the tool advances per revolution of work.

Digitized by Google

94. The cut-meter, *A*, Fig. 76, may be used to measure cutting speed automatically, also speed of drills, milling cutters, etc. It consists of a case *B*, which contains the magnetic mechanism for registration. Scale *C* is calibrated to read the

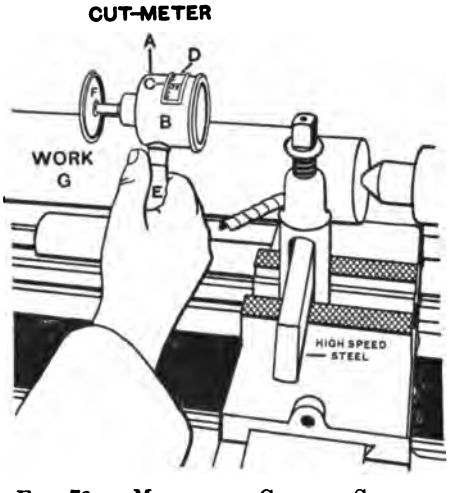


FIG. 76. — MEASURING CUTTING SPEED.

cutting speed in feet per minute; the 0 line is on glass *D*. To use the cut-meter, it is held by handle *E* and wheel *F* pressed against the revolving work *G*.

95. Surface speed attachment for speed indicator. — A rubber-tired wheel *A*, Fig. 77, 6″ in circumference, is slipped over point of a speed indicator. See *Elements of Machine Work*. The wheel is pressed against work as at *B*, Fig. 75, and number of revolutions noted in a given time, as one minute. To get surface speed in feet, multiply number of revolutions per minute by 2.

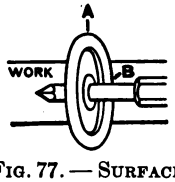


FIG. 77. — SURFACE SPEED ATTACHMENT FOR SPEED INDICATOR.

96. To find the lathe revolutions, given the cutting speed and diameter of the work: Multiply the cutting speed by 12 to reduce it to inches, then divide by the diameter of the work multiplied by 3.1416.

Digitized by Google

Example. — The work is 2″ in diameter and it is desired to turn it at 35 feet per minute. How many lathe revolutions are necessary?

Solution. $$\frac{35 \times 12}{2 \times 3.1416} = 67 \text{ R.P.M.}$$

97. To find the cutting speed, given diameter of the work and lathe revolutions per minute: Multiply diameter of work by 3.1416 and by number of lathe revolutions, then divide by 12.

Example. — The work is 2″ in diameter and makes 67 revolutions per minute in the lathe. What is the cutting speed?

Solution. $$\frac{2 \times 3.1416 \times 67}{12} = 35 \text{ feet.}$$

98. Eight or ten speeds are possible in an engine lathe. — Place the belt on step of cone that will give the speed nearest to that required.

TABLE OF LATHE CUTTING SPEEDS.

Materials.	Speeds for Roughing.	
	Carbon Steel Tools.	High-speed Steel Tools.
Cast iron	30	60
Steel or wrought iron	25	50
Carbon steel, annealed	20	40
Brass composition	95	190

Speed for finishing is 50% to 100% higher than roughing speed.

Speed for filing (brass excepted) equals four times roughing speed.

Speed for filing brass equals three times roughing speed.

Attention. — Roughing cuts are not to reduce diameters more than $\frac{1}{8}$″; and finishing cuts, not more than $\frac{1}{8}\frac{1}{4}$″. Roughing feeds are not to be less than 17 revolutions per 1″ tool

Digitized by Google

travel, and finishing feeds are not to be less than 90 revolutions per 1″ tool travel.

If the above is exceeded, a less speed than that given in table must be used.

For large diameters, use reduced speed, as the strain on the tool is greater.

99. Cutting feeds. — Lathes with belt feed are limited to three changes; lathes with gear feed are not limited.

For heavy work, coarse feed is about 17 lathe revolutions to 1″ of tool travel.

For average work, medium feed is about 38 lathe revolutions to 1″ of tool travel.

For average work, fine feed is about 90 lathe revolutions to 1″ of tool travel.

For tool making, a feed of 200 revolutions is often used.

Attention. — A student may use finer feeds until he acquires some experience; about 80 to 1″ for roughing and 140 to 1″ for finishing.

LUBRICANTS FOR CUTTING TOOLS.

100. Some metals are machined dry, others require a lubricant. — Cast iron, with about two exceptions, as tapping and polishing, must be machined dry. Steel and wrought iron, with two exceptions, as machinery with side or diamond-point tool (and these are optional), must be machined with a lubricant. A neglect of this may cause destruction of both tool and work. See Table of Lubricants, p. 51.

A lubricant on tenacious metal prevents excessive friction and conducts away heat, thus preserving the point of the tool and producing a smooth finish on the work. It also helps to carry away the chips.

Attention. — Preferably use sperm oil or lard oil on Arkansas oilstones. Use kerosene oil or lard oil on India oilstones.

Oil. — "Oil" in tables means lard oil or some oil mixture. Mixtures and compounds are often used as substitutes because of cheapness.

Note. — Never use mineral oils for cutting tools.

Digitized by Google

TABLE OF LUBRICANTS FOR CUTTING TOOLS.

Operations.	Metals.						
	Cast Iron	Machine Steel or Wrought Iron.	Carbon or High-Speed Steel.	Copper.	Brass or Bronze.	Aluminium.	Lead. Babbitt.
Turning. Boring.	Dry.	Dry, oil or soda water.	Dry or oil.	Milk.	Dry.	Kerosene or turpentine.	Dry.
Cutting off. Grooving.	Dry.	Oil or soda water.	Oil or soda water.	Milk.	Dry.	Kerosene or turpentine.	Dry.
Screw cutting.	Dry.	Oil.	Oil.	Milk.	Dry.	Kerosene or turpentine.	Dry.
Threading with dies				Milk.	Dry.	Kerosene or turpentine.	Oil.
Tapping.	Oil.	Oil.	Oil.				
Drilling. Countersinking. Counterboring.	Dry.	Oil or soda water.	Oil or soda water.	Milk.	Dry.	Kerosene or turpentine.	Dry.
Chucking.	Dry.	Oil or soda water.	Oil or soda water.	Milk.	Dry.	Kerosene or turpentine.	Dry.
Reaming.	Dry.	Oil.	Oil.	Milk.	Dry.	Kerosene or turpentine.	Dry.
Milling.	Dry.	Oil, soap mixture, or soda water.	Oil, soap mixture, or soda water.	Milk.	Dry.	Kerosene or turpentine.	Dry.
Planing.	Dry.	Dry, oil or soda water.	Dry or oil.	Milk.	Dry.	Kerosene or turpentine.	Dry.
Nurling.	Oil.	Oil.	Oil.	Milk.	Oil.	Kerosene or turpentine.	Dry
Filing.	Dry.	Dry or oil.	Dry or oil.	Dry or milk.	Dry.	Kerosene or turpentine.	Oil.
Polishing with emery cloth.	Oil.	Oil.	Oil.	Oil.	Oil.	Oil.	Oil.

Digitized by Google

Soda water (sal soda dissolved in water) is useful in machining steel or wrought iron, and produces what is known as a water finish.

Soap mixture. — ½ lb. sal soda, ½ pt. lard oil, ½ pt. soft soap, 10 qts. water; boil one-half hour. This is good for milling and drilling steel and wrought iron.

101. Drilling extra hard steel such as unannealed carbon steel. — Use a flat drill, with turpentine or kerosene oil as a lubricant. See § 445.

102. Drilling glass. — Use a very hard flat drill, scratch glass with an old file to remove polish and use turpentine as a lubricant. Drill part way through, then turn over and finish from the other side to avoid chipping surface.

INSPECTING AND MEASURING STOCK. OILING AND CLEANING MACHINES.

103. Inspecting and measuring stock (material). — On receiving a piece of stock, inspect it for imperfections, as blowholes in castings, flaws (cold short) in forging or bars. Stock for work to be hardened must be carbon or high-speed steel. To distinguish high-grade machine steel from tool steel, a trial piece may be tested by the hardening process.

Measure piece to see if it is large enough to finish to dimensions given on the drawing. If the piece be a rough casting or forging, there should be at least ⅛″ surplus stock; if smooth, $\frac{1}{16}$″.

When requested to finish work begun by another, inspect and measure it, and make a note of its condition, that you may not be held responsible for errors not your own.

104. Rough turn all over before finishing. — As metals alter in form when the skin or outside is removed, when possible remove the skin from all surfaces before finishing any part. An exception is made in some classes of lathe work, as a shaft which is squared to exact length before the diameter is roughed out.

Digitized by Google

105. Care of machines, small tools, and benches. — When through using a machine, clean it, first with a brush, then wipe with cotton waste.

Wrenches, handles, bolts, straps, and fixtures should be put away in their proper places so that they may be found when wanted. All tools out on checks should be returned to the tool room as soon as possible.

The bench and vise may be cleaned with a brush. Files and other tools should be arranged so that they will not be injured by marring each other and that they may be readily reached. Be *neat, aggressive,* and *self-reliant.*

106. Lathe box or tray for tools and work. — Do not place tools, work, or other metallic objects on the ways of a machine, as they would scar and affect their truth. A wooden box or tray should be supplied for the tools or small work. It may be placed on the ways near the end of the lathe. Large work should be placed on a bench, truck, or the floor.

Digitized by Google

CHAPTER V.

ENGINE-LATHE WORK. CENTERING, SQUARING, AND STRAIGHT TURNING. FILING LATHE WORK. MICROMETER AND VERNIER CALIPERS.

ENGINE-LATHE WORK.

107. Schedule of operations, indicated on drawings of machine parts. — To avoid performing work in an improper order, plan a schedule of operations, machines, and tools for one or duplicate parts, from the rough stock to the finished piece. Indicate the operations on drawing by numbers or letters.

A schedule for students should be very complete.

108. A multiple schedule for two or more duplicate pieces is the same as for a single piece, except that each operation is performed on all pieces before beginning the next.

109. The student in machine construction may begin lathe work by turning soft cast iron, Fig. 78, as it machines more easily than steel or wrought iron; the cutting angles of the tools are easy to shape, and the material is not expensive.

110. To mount work on lathe centers. — Clean center holes and centers with waste. Fasten dog on work (see Fig. 15), mount on live center. With left hand under end of work, hold work in line with dead center; move and clamp foot-stock width of tool block from end of work. Oil dead center. With the little finger of left hand to guide the dead center into center hole, screw out spindle with the right hand until there is no end movement. To test adjustment, move tail of dog back and forth; when right a slight resistance is felt by the hand; clamp binder.

Digitized by Google

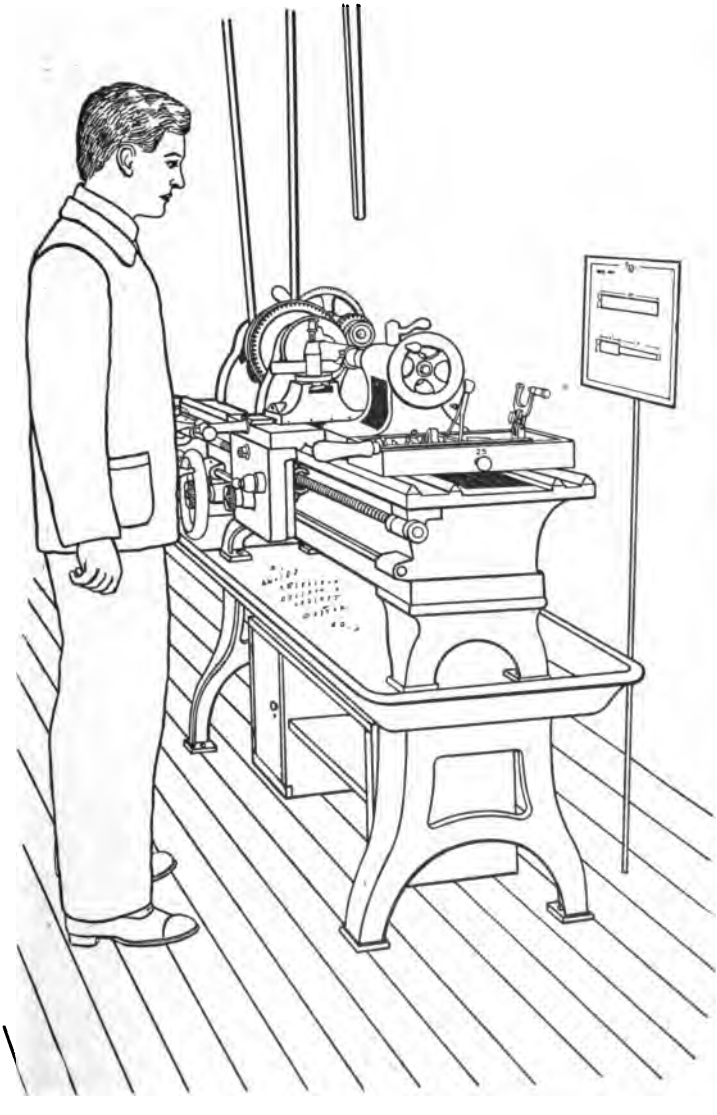


FIG. 78. — CORRECT POSITION AT ENGINE LATHE FOR TURNING.

Digitized by Google

Warning. — When taking a heavy cut or rotating work at a high speed, the work will heat and expand and thus bind on the centers. The student should be watchful and relieve and oil the dead center occasionally. If this is neglected, a *"hot center" will result, which usually destroys both work and center.*

111. To turn work to one diameter from end to end. — First, set tool to take roughing cut and turn approximately one-half the length (Fig. 79). Stop feed, then stop lathe.

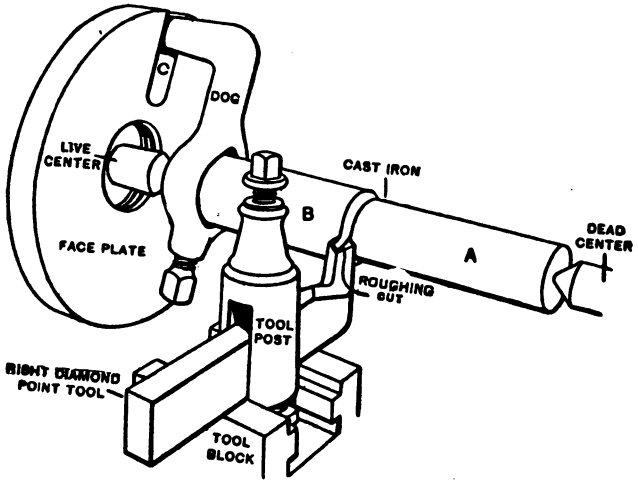


FIG. 79. — ROUGH TURNING SHAFT FROM END TO END, FIRST HALF.

Take stock out of lathe and run carriage back by hand to dead center. Do not disturb cross feed. Fasten dog on turned end *A'*, Fig. 80, with a piece of copper *D* under set screw to avoid scarring work. Again mount work on centers and turn off half marked *B'*. Regrind tool and follow same method with finishing cut.

Digitized by Google

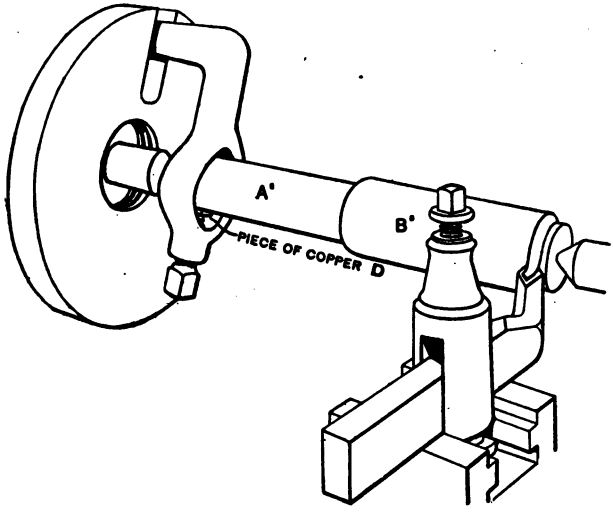


FIG. 80. — ROUGH TURNING SHAFT FROM END TO END, SECOND HALF.

CENTERING, SQUARING AND STRAIGHT TURNING.

112. To center, square and turn straight. See Figs. 81 and 82.

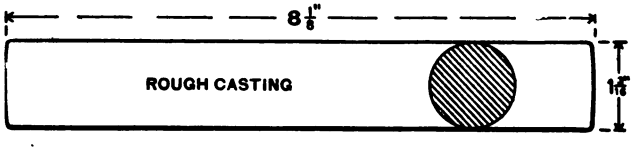


FIG. 81.

SCHEDULE OF OPERATIONS, MACHINES, AND TOOLS.

ROUGHING. FINISHING.

Stock, iron casting $\frac{1}{8}$" large, Fig. 81, free from visible defects.

Oil bearings of lathe with machine oil.

True live center. Set dead center in accurate alinement.

Use carbon-steel cutting tools. See *Exception*, p. 59.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES, AND TOOLS.

Concluded.

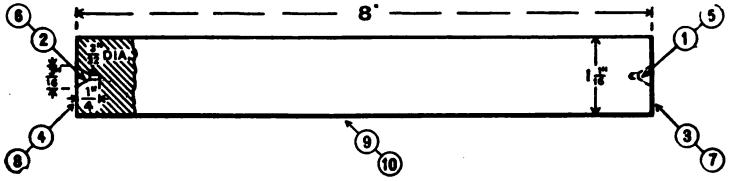


FIG. 82. — SCHEDULE DRAWING OF CENTERING, SQUARING AND STRAIGHT TURNING.

OPERATIONS.	MACHINES, SPEEDS, FEEDS.	TOOLS.
Snag. Center by hand method, §46. Center holes $\frac{3}{16}$″ diameter, (1), (2).	Speed lathe 8″ to 12″. 3d or 4th speed, or 1400 R. P. M.*, or countersink speed 700 R.PM.	Vise, hammer, chisel, file, chalk, dividers, rule, center punch, $\frac{3}{32}$″ drill, 60° countersink.
Mount and adjust on centers. Rough square (3), round nose inward, heavy cut, and side tool outward, light cuts. Reverse work on centers and repeat on (4). Test length with calipers. Take light cuts outward on (4) until work measures 8″ + $\frac{1}{64}$″.	Engine lathe 12″ to 16″. 2d speed, or 40 F.P.M.† Hand feed.	Dog, round-nose and side tools, 15° rake, calipers, 12″ steel rule.
Recenter to $\frac{3}{16}$″, (5), (6).....	Speed lathe.	60° countersink.
Regrind and oilstone tool and finish square (7) one cut. Reverse work on centers and repeat on (8). Test length with calipers. Take light cuts on (8) until work measures 8″. To remove burr around countersinks, see §§ 68, 69. Test flatness of end with steel rule.	Engine lathe. 3d speed, or 60 F.P.M. Hand feed.	Side tool, 15° rake, calipers, rule.
Rough turn to $1\frac{1}{16}$″ + $\frac{1}{64}$″ one cut. See Figs. 79 and 80, § 111 (9). Test with calipers at ends and middle.	1st or 2d speed, or 25 to 40 F.P.M. Power feed 80 to 1″.	Copper under set screw of dog, diamond-point tool, 15° rake, small calipers, 3″ steel rule.
Regrind and oilstone tools. Finish turn to $1\frac{1}{16}$″, § 76, (10) one cut. Test with micrometer calipers, at both end and middle. Limit allowed .001″. Stamp name on end.	3d speed, or 55 F.P.M. Fine power feed 140 o 1″.	Diamond-point tool or round-nose tool, 15° rake. Calipers, micrometer. Name stamp.
Clean lathe.		Brush and waste.

* R.P.M. = revolutions per minute. † F.P.M = feet per minute.

Digitized by Google

Attention. — If necessary, when roughing the tool may be removed and reground, but in finishing it should be prepared to carry its cut without regrinding.

Note. — When squaring, hold long. feed handle firmly with one hand while operating cross feed handle with the other.

Caution. — As a heavy cut may draw the tool inward, caliper occasionally and readjust tool, if need be, to avoid roughing diameter too small.

Exception. — If a schedule says "use carbon-steel cutting tools" and it is desired to change to "use high-speed steel," the cutting speed may be increased from 50% to 100%.

If a schedule says "use high-speed steel cutting tools" and it is desired to change to "carbon-steel" the cutting speed *must* be reduced 50% to 100%.

FILING LATHE WORK.

113. Lathe work is filed to remove tool marks, to make a fit, to produce an exact diameter, and also to prepare the surface for polishing. A small amount of filing improves the condition of the work; excessive or careless filing will destroy its truth.

114. Mill files are best for lathe work, files single-cut (sometimes called floats), as at *C*, Fig. 83. A mill bastard is useful for a large variety of lathe filing, but a mill 2d cut is only used for finer classes of work.

115. Speed for filing. — If work is revolved at too high a speed, the file will not bite, but will simply glaze the work and rapidly destroy itself. As a rule, in filing steel and cast iron the lathe may be run at a speed between four and five times faster than that used in taking the roughing cut. Cast iron must be filed dry. See Table of Cutting Speeds, § 98. On brass, the rule is from two to three times the roughing speed.

On work $1\frac{1}{2}''$ and less in diameter, run engine lathe at its fastest speed. Small work $\frac{1}{2}''$ and less may be filed in the speed lathe.

Digitized by Google

To avoid a hot center, loosen the dead center and oil it before speeding up for filing.

116. To hold and to use a file on lathe work. — The work *A*, Fig. 83, is revolved at a moderately fast speed in the direction of arrow *B*, while mill file *C* is moved in the direction of arrow *D*. The stroke should be slower than in vise work, in

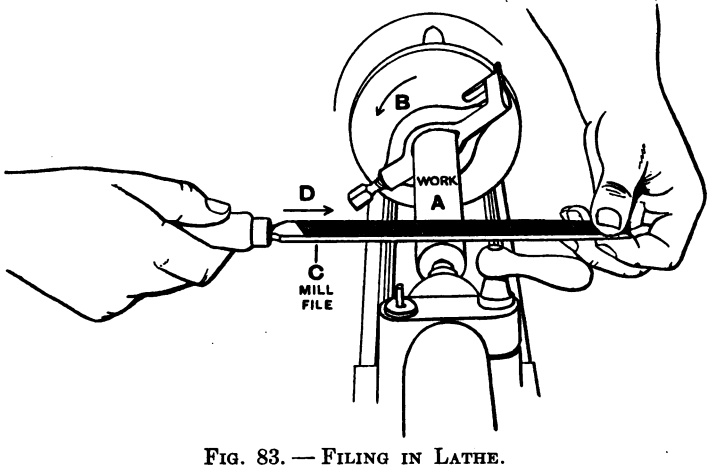


FIG. 83. — FILING IN LATHE.

order that the work may make a number of revolutions during each stroke; the pressure should be lighter, as the number of teeth in contact are fewer.

Attention. — Clean the file frequently with a file card to remove chips, or they will scratch the work.

117. To finish radial or side surfaces. — A hand file is sometimes used, but the better method is to use a scraper. See § 313.

118. Round and half-round files for lathe work. — Fillets and concave surfaces are generally made with forming tools and scraped with hand tools. When it is desired to polish such surfaces, they may be prepared by filing with round or half-round files, giving the file on its forward stroke a slight sweep following the curve.

See Lubricants for Cutting Tools, §100.

Digitized by Google

MICROMETER CALIPERS.

119. The micrometer principle consists of a combination of an accurate screw and graduated head or nut by which fine measurements and adjustments may be obtained. It is applied to feed screws of lathes, planers, milling and grinding machines, etc., and to instruments of precision, as the micrometer caliper, Fig. 84, which consists of frame *A* and barrel *B*;

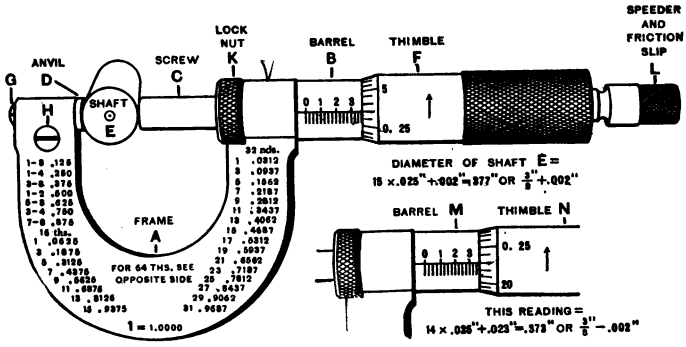


FIG. 84. — MEASURING WITH MICROMETER CALIPER.

in the end of the barrel is a fixed nut through which passes screw *C*. The caliper is graduated to read in thousandths of an inch, but half, quarter, and tenths of thousandths may be readily estimated.

120. To measure work with micrometer. — Place shaft *E* between the faces. Rotate thimble *F* with thumb and finger until a light but distinct contact is obtained. Prove results by moving micrometer up and down on shaft *E*, or by passing work between micrometer anvil *D* and screw *C* with a slight rotative motion.

121. To read micrometer. — Screw *C* has 40 threads to the inch; the graduations on barrel *B* are 40 to the inch. One revolution of the screw opens the caliper one-fortieth of an inch, or .025″. Thimble *F* is graduated into twenty-five parts. Each division when passing the axial line on barrel indicates one-twenty-fifth of one-fortieth of an inch ($\frac{1}{25} \times \frac{1}{40} = \frac{1}{1000}''$).

Digitized by Google

Reading. — Every fourth division on the barrel is figured 1, 2, 3, 4, etc., and may be read 0.100″, 0.200″, etc.; the figure 3 may be read 0.300″ and the three additional divisions as 0.075″, making 0.375″ on barrel. Then add the two divisions or 0.002″ on the thimble, which makes the complete measurement 0.377″.

Attention. — A student occasionally reads the example at *MN* backward from zero and obtains .377″ instead of .373″.

122. Adjusting the anvil to correct error. — If zero lines do not coincide when anvil and screw are clean and in contact, adjust anvil *D* by screw *G*, first loosening screw *H*.

123. Lock nut *K* may be used to clamp the screw and preserve any setting.

124. Speeder and friction slip consists of a ratchet and pawl used as a speeder for the rapid movement of screw, and as a friction slip so that the same pressure of contact may be obtained at every reading.

125. A one-hand method of measuring with a micrometer. Fig. 85. — Hold work *A* with left hand and insert third finger

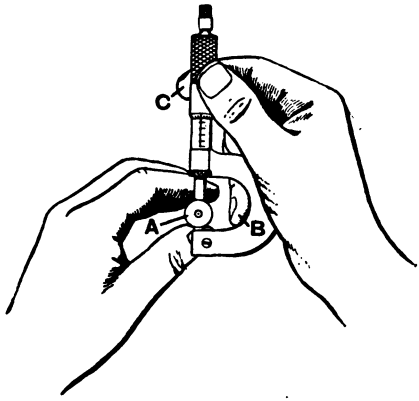


FIG. 85. — MEASURING WORK WITH MICROMETER HELD WITH ONE HAND.

of right hand into frame to steady caliper *B*. Adjust screw *C* to work by rotating the thimble with the thumb and first finger.

Digitized by Google

126. To measure work in a lathe with micrometer. Fig. 86. — Hold frame *A* over work *B* and operate thimble *C*.

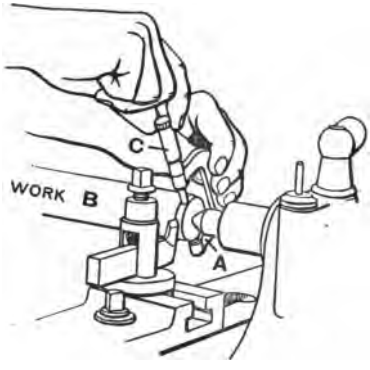


FIG. 86. — MEASURING WORK IN THE LATHE WITH MICROMETER.

127. Large micrometer and stand. — Micrometer stand *A*, in Fig. 87, on bench *B*, not only protects micrometer *C* from

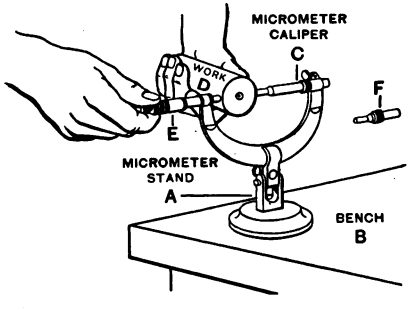


FIG. 87. — MEASURING WORK ON BENCH WITH MICROMETER.

injury, but avoids expansion due to the heat of the hand. To measure, hold work *D* as shown. This micrometer is of the interchangeable anvil type. One of the shorter anvils is shown at *F*.

Digitized by Google

Standard end-measuring rods and disks are supplied with micrometers measuring more than one inch, to test their accuracy.

128. Decimal equivalents of common fractions. — As micrometers are graduated to read decimally, the common fractions and their decimal equivalents are stamped on the frames. A student should memorize the decimal equivalents of such fractions as $\frac{1}{2}''$, $\frac{1}{4}''$, $\frac{1}{8}''$, $\frac{1}{16}''$, $\frac{1}{32}''$, and $\frac{1}{64}''$.

TABLE OF COMMON FRACTIONS AND DECIMAL EQUIVALENTS.

	$\frac{1}{64}$				.015625			$\frac{17}{32}$			.53125
		$\frac{1}{32}$			.03125						
	$\frac{3}{64}$				.046875		$\frac{35}{64}$				.546875
			$\frac{1}{16}$		.0625						
	$\frac{5}{64}$				.078125				$\frac{9}{16}$		.5625
		$\frac{3}{32}$			.09375						
	$\frac{7}{64}$				.109375		$\frac{37}{64}$				.578125
				$\frac{1}{8}$	.125						
	$\frac{9}{64}$				.140625			$\frac{19}{32}$			.59375
		$\frac{5}{32}$			.15625		$\frac{39}{64}$				.609375
	$\frac{11}{64}$				.171875					$\frac{5}{8}$	.625
			$\frac{3}{16}$		.1875		$\frac{41}{64}$				.640625
	$\frac{13}{64}$				.203125			$\frac{21}{32}$			.65625
		$\frac{7}{32}$			.21875		$\frac{43}{64}$				.671875
	$\frac{15}{64}$				.234375				$\frac{11}{16}$		.6875
				$\frac{1}{4}$	.250		$\frac{45}{64}$				.703125
	$\frac{17}{64}$				.265625			$\frac{23}{32}$			.71875
		$\frac{9}{32}$			.28125		$\frac{47}{64}$				.734375
	$\frac{19}{64}$				.296875					$\frac{3}{4}$	.750
			$\frac{5}{16}$		.3125		$\frac{49}{64}$				.765625
	$\frac{21}{64}$				.328125			$\frac{25}{32}$			.78125
		$\frac{11}{32}$			.34375		$\frac{51}{64}$				.796875
	$\frac{23}{64}$				.359375				$\frac{13}{16}$		.8125
				$\frac{3}{8}$	.375		$\frac{53}{64}$				.828125
	$\frac{25}{64}$				.390625			$\frac{27}{32}$			.84375
		$\frac{13}{32}$			.40625		$\frac{55}{64}$				.859375
	$\frac{27}{64}$				.421875					$\frac{7}{8}$	.875
			$\frac{7}{16}$		.4375		$\frac{57}{64}$				.890625
	$\frac{29}{64}$				.453125			$\frac{29}{32}$			.90625
		$\frac{15}{32}$			.46875		$\frac{59}{64}$				.921875
	$\frac{31}{64}$				.484375				$\frac{15}{16}$		.9375
				$\frac{1}{2}$	.500		$\frac{61}{64}$				.953125
								$\frac{31}{32}$			.96875
	$\frac{33}{64}$				.515625		$\frac{63}{64}$				.984375
										1	1.000000

Digitized by Google

VERNIER CALIPERS.

129. Vernier principle. — Some instruments of precision have in conjunction with the main rule (or scale) a short movable scale, called a vernier, to read fractional parts of the smallest divisions on the main rule.

The vernier is divided into one more or one less divisions than a given number of divisions on the rule. A division on the rule thus differs from a division on the vernier by the fraction shown by 1 divided by the number of divisions on vernier.

Rule *A*, Fig. 88, is divided into inches, tenths, and fortieths ($\frac{1}{40}'' = .025''$). Vernier *B* is divided into twenty-five

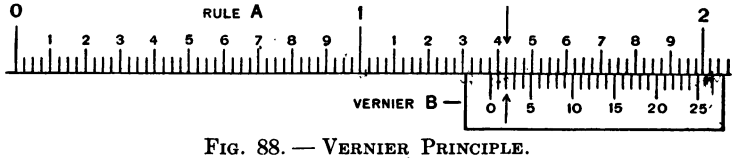


FIG. 88. — VERNIER PRINCIPLE.

divisions which equal twenty-four divisions on Rule *A*. As each division on rule *A* is .025″ ($\frac{1}{40}''$), each division on Vernier *B* is .024″ or .001″ less ($\frac{1}{40}'' \times \frac{1}{25}'' = \frac{1}{1000}'' = .001''$).

If the zero line on vernier is set to coincide with the zero line or any line on rule *A*, the next two lines to the right will differ from each other by .001″, and the difference will increase .001 for each division.

130. Example in reading. — *First, read the rule.* Each tenth is read .100″ and each fortieth .025″. The first line on rule to left of zero line on vernier is distant from zero line on rule .300″ and three fortieths, .075″, which gives 1.375″.

Second, read vernier. Advancing to the right, we find that the line at the end of the second division on the vernier coincides with a line on the rule as indicated by arrows, .002″, is the distance of the zero line on the vernier from the division on the rule.

Third, to reading of rule add reading of vernier to obtain complete measurement. Thus: 1.375″ + .002″ = 1.377″.

Digitized by Google

131. The application of vernier principle to the vernier caliper in Fig. 89. Each inch of beam *C* is divided into 40 parts, and vernier *D*, attached to sliding head *E*, into 25 parts. To measure work, as shaft *F*, the caliper is placed over the shaft and the sliding head is moved to bring sliding jaw *G* and solid jaw *H* in contact with shaft, clamp *K* is fastened to beam by thumb screw *L*. The jaws are made to touch shaft delicately yet distinctly by adjusting nut *M*, after which head *E* is fastened to the beam

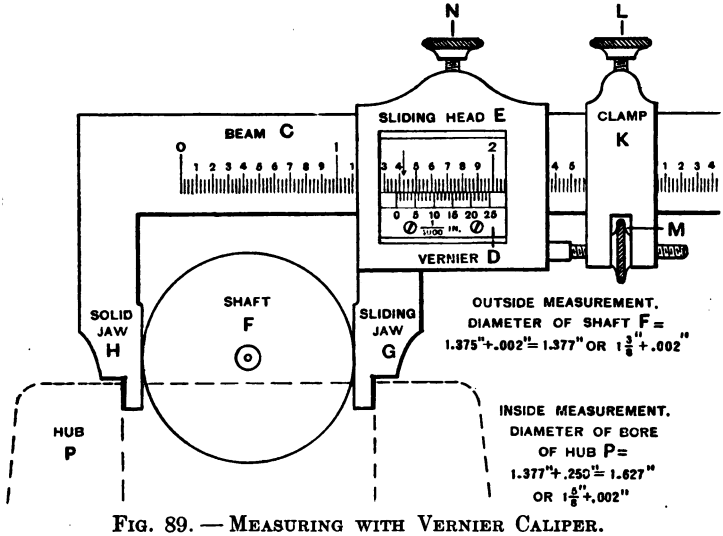


FIG. 89. — MEASURING WITH VERNIER CALIPER.

by thumb screw *N*. The caliper reads the same as the former example, Fig. 88, that is: 1.377″ or $1\frac{3}{8}$″ + .002″. It is best to use a magnifying glass to read or set a vernier caliper.

132. To obtain inside measurements with a vernier caliper. — To take inside measurements, as the bore of hub *P* shown dotted, the caliper is read exactly as above, then the width of the points of the jaws, which differ with the size of the caliper is added. For the caliper in Fig. 89 two hundred and fifty thousandths of an inch (0.250″) must be thus added to the reading on the vernier side.

Digitized by Google

133. Vernier caliper as a caliper square. — The vernier caliper may be used as a caliper square for ordinary outside and inside measurements by reading the back of beam *C*, which is graduated to read to 64ths.

134. A ten-thousandth micrometer. — Micrometer calipers are obtainable to read to the tenth part of a thousandth of an inch. A vernier of ten divisions is marked on the barrel *A*, Fig. 90, and in the space occupied by nine divisions on the

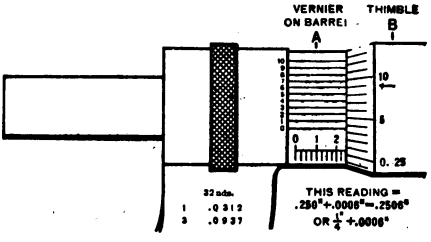


FIG. 90. — A TEN-THOUSANDTH MICROMETER.

thimble *B*. The micrometer reads .250″+. To read the vernier to obtain the fourth decimal place, locate the line on the vernier, as line 6, which coincides with a line on the thimble, and add .0006″ to reading, as .250″ + .0006″ = .2506″ or ¼ + .0006″.

Digitized by Google

CHAPTER VI.

FITS IN MACHINE CONSTRUCTION WITH TABLES OF ALLOWANCES. FORCING PRESS. STANDARD AND LIMIT GAGES.

FITS IN MACHINE CONSTRUCTION WITH TABLES OF ALLOWANCES.

135. Fits in machine construction are most important, and unless made according to requirements, impair the usefulness of the machine. See Dimension-Limit System, *Elements of Machine Work.*

Examples.—In an engine lathe the live spindle of steel is a running fit in bronze or Babbitt boxes in the headstock. The footstock spindle of steel is a sliding fit in the cast-iron footstock. The headstock cone is a running fit and the headstock gear a drive fit on spindle. The headstock, footstock, and carriage of cast iron are sliding fits on the ways of the cast-iron bed. See Scraping, *Elements of Machine Work.*

Like metals are generally used for sliding surface fits, *unlike* metals for running fits. Cast iron to cast iron wears well for any fit, but steel to steel will quickly abrade unless hardened and ground.

136. The classes of fits used in machine construction are: running, sliding, driving, forcing and shrinking fits; and taper fits, running and forcing. The hole or bore for duplicate work should be standard. See § 150.

137. Running fits vary with the class of work. The amount of looseness varies from .0002″ on fine watch work to $\frac{1}{64}$″ on some classes of cotton and woolen machinery. See Tables, Classes I, II, III, page 69. Running fits are often made taper. The adjustment is obtained by moving the spindle along the box, or the box along the spindle.

Digitized by Google

Running Fits. — Class I.

TABLE OF ALLOWANCES AND LIMITS UNDER STANDARD FOR FINE WORK.

Hole Diameter, Inches.	Shaft Diameter = Hole Diameter —	Limit Shaft Diameter + or —
.00 to .49	.000625	.000125
.50 to 1.99	.0012	.000375
2.00 to 3.99	.0016	.000625
4.00 to 5.99	.002	.00075
6.00 to 8.00	.002375	.000875

Running Fits. — Class II.

TABLE OF ALLOWANCES AND LIMITS UNDER STANDARD FOR AVERAGE WORK WHERE HIGH SPEEDS ARE REQUIRED.

Hole Diameter, Inches.	Shaft Diameter = Hole Diameter —	Limit Shaft Diameter + or —
.00 to .49	.001	.00025
.50 to .99	.0015	.0005
1.00 to 1.99	.0019	.000625
2.00 to 2.99	.0023	.00075
3.00 to 3.99	.0027	.00075
4.00 to 5.99	.0035	.001
6.00 to 8.00	.0038	.00125

Running Fits. — Class III.

TABLE OF ALLOWANCES AND LIMITS UNDER STANDARD FOR ENGINE WORK WHERE EASY FITS ARE REQUIRED.

Hole Diameter, Inches.	Shaft Diameter = Hole Diameter —	Limit Shaft Diameter + or —
.00 to .49	.0015	.0005
.50 to .99	.002	.00075
1.00 to 1.99	.0026	.000875
2.00 to 3.99	.0034	.001125
4.00 to 5.99	.00425	.00125
6.00 to 8.00	.005	.0015

Digitized by Google

138. Sliding fits such as the footstock spindle of lathes are made similar to running fits except that the final fitting is done by draw-filing. See *Elements of Machine Work.*

Many lathe manufacturers grind the spindle slightly large, then force it back and forth in the footstock with a power press, which smooths, straightens, and stretches the hole, eliminates wear to some extent and produces a fine sliding fit.

TABLE OF ALLOWANCES AND LIMITS UNDER STANDARD FOR SLIDING FITS.

Hole Diameter, Inches.	Shaft Diameter = Hole Diameter −	Limit Shaft Diameter + or −
.00 to .49	.0005	.00025
.50 to .99	.00075	.00025
1.00 to 1.99	.00125	.00025
2.00 to 3.99	.00175	.00025
4.00 to 5.99	.00225	.00025
6.00 to 8.00	.0035	.0005

139. Driving or drive fits, easy and hard are made by turning the shaft to size with allowance for filing. For easy fits, file or grind until shaft will enter hole about two-thirds length of fit with hand pressure; for hard fits, one-third. The former are used for light-keyed fits and small work; the latter for ordinary work which may be removed for repair.

TABLE OF ALLOWANCES AND LIMITS OVER STANDARD FOR DRIVING FITS.

Hole Diameter, Inches.	Shaft Diameter = Hole Diameter +	Limit Shaft Diameter + or −
.00 to .49	.000375	.000125
.50 to 1.24	.00075	.00025
1.25 to 2.49	.00125	.00025
2.50 to 8.00	.002	.0005

Digitized by Google

140. Forcing or force fits. — In assembling and erecting machinery, nothing is more important than the proper fitting of the parts that have to be driven or forced together, or driven or forced apart, when making repairs. Examples of forcing fits are gears, couplings, locomotive driving wheels, crank pins, car axles, rod bushing, crank disks, various kinds of bushings, linings into cylinders, various parts of engines, generators and motors, iron bands on wagon-wheel hubs, crank shafts into automobile fly wheels, various parts of built-up cranks, or any two machine parts that have to be joined by forcing one into the other with sufficient power to prevent them ever becoming loose.

TABLE OF ALLOWANCES AND LIMITS OVER STANDARD FOR FORCING FITS.

Hole Diameter, Inches.	Shaft Diameter = Hole Diameter +	Limit Shaft Diameter + or −
.00 to .49	.00075	.00025
.50 to .99	.0015	.0005
1.00 to 1.49	.0025	.0005
1.50 to 1.99	.0035	.0005
2.00 to 2.49	.0045	.0005
2.50 to 3.24	.0055	.0005
3.25 to 3.99	.0065	.0005
4.00 to 4.99	.0075	.0005
5.00 to 5.99	.0085	.0005
6.00 to 8.00	.0095	.0005

Attention. — To avoid abrasion and destruction of work, lubricate both surfaces with machine oil. For heavy work use cylinder oil, white lead, or grease.

141. Taper forcing fits. — For some classes of machinery, such as marine and engine work, taper forcing fits are used. The hole is bored to a taper of $\frac{1}{16}''$ to 1″, and the shaft ground or turned the same or to a slightly greater taper, and from .010″ to .020″ larger, then forced in.

On some classes of work, the hole is made straight and the shaft ground or turned to a taper of about .001″ to 1″, which makes an effective fit.

Digitized by Google

142. Pressures and allowances for forcing fits. — Formulas have been deduced and tables made giving pressures required to force two parts together; these possess considerable value, but serve only as a guide, for the pressure required depends not only on the difference in diameters but on the diameter and length of fit, smoothness of both parts, material of both shaft and hub, and diameter of hub. The material will often make a difference of from twenty-five per cent to fifty per cent in the pressure for the same allowance. For this reason the method that is generally followed is to make tables of allowances and pressures for each class of fits and for different materials.

FORCING PRESS.

143. Forcing presses for forcing fits. — The old method of making forcing fits is with sledge hammers, rams, etc., but such methods are now nearly obsolete except for occasional fits, as the process is difficult, dangerous, and lacks uniformity. The economical, easy, safe, and scientific method is by hydraulic-power or belt-power presses of which there are a great variety both vertical and horizontal, small and large, stationary and portable, to suit all classes of work. Small forcing fits may be made with a mandrel or arbor press.

144. To force screw into bevel gear. Fig. 91. **Belt-power forcing press.** — This type of press may be used for forcing fits between mandrel press work and hydraulic press work and not requiring a pressure exceeding fifty tons.

Screw *A* is keyed, lubricated, preferably with linseed oil, and forced into bevel gear *B* by ram *C*, to which motion is transmitted by gearing actuated by the driving belt *D* and controlled by hand wheel *E* which operates a friction. The hand wheel *E* is used to raise and lower the ram and to control the pressure through its operation of a friction device; the harder the wheel is turned the greater the friction and the greater the pressure produced. The friction only acts on the downward motion.

The plunger *F* which compresses glycerine in a chamber at the end of ram *C*, records the pressure in tons per square inch by means of gage *G*.

Digitized by Google

Specifications. — The gear in Fig. 91 is steel case-hardened; hole straight, $1\frac{1}{4}''$ diameter; screw, crucible steel; length of fit, 3″; allowance, and taper, 1.250″ in diameter at end and 1.254″ at shoulder (ground); pressure, 8 tons.

145. To force shaft into flange, forcing fit. Fig. 92. **Belt-power forcing press.** — Shaft *A* is keyed, oiled, and forced into flange *B*.

Specifications. — Flange, cast iron; hole 1″ diameter; shaft, machine steel; length of fit, $1\frac{3}{4}''$, straight; allowance, .002″ to .0025″, large (ground); pressure, about 3 tons.

Attention. — Condition of surfaces (smooth or rough) will vary the pressure.

146. To force shaft into malleable-iron gear.

Specifications. — Malleable-iron gear; hole $1\frac{3}{16}''$ diameter; length of fit, 2″; shaft, crucible steel; allowance and taper, 1.1875″ and 1.1915″ at shoulder; pressure, 5 tons.

147. To force shaft into machine-steel gear.

Specifications. — Machine-steel gear; hole 2″ diameter; length of fit, 5″; shaft-machine steel; allowance and taper, 2.000″ at end, 2.004″ at shoulder; pressure, 13 to 15 tons.

148. Shrinking fits are used to fasten a collar, sleeve, crank pin, crank, or other piece permanently in place. They differ from forcing fits in manner of fitting only. The fit is made by heating the hollow piece until it expands sufficiently to go on the cold shaft easily. The parts must be put together quickly or the heat will expand the shaft and the parts stick hard before they are in place, in which case they should be driven or pressed apart as quickly as possible. When in place, cool slowly with water.

If the proper shrinkage is allowed, it will not be necessary to heat piece above a dull red, 800° F.

For small work the hole may be standard and the amount for shrinkage allowed on shaft.

Digitized by Google

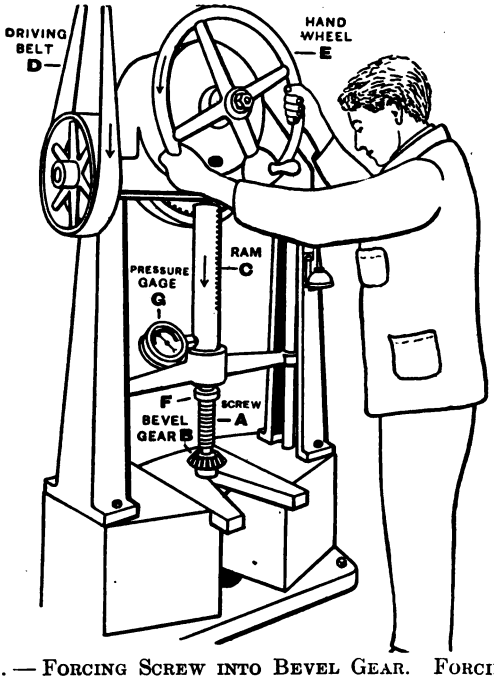


FIG. 91. — FORCING SCREW INTO BEVEL GEAR. FORCING FIT. BELT-POWER FORCING PRESS.

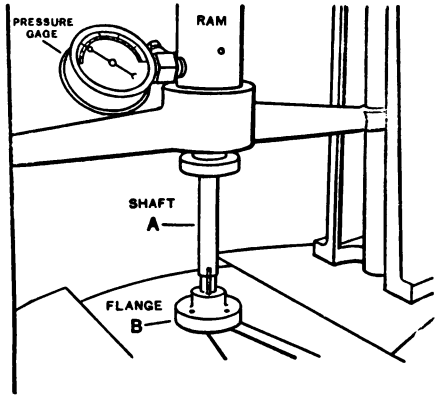


FIG. 92. — FORCING SHAFT INTO FLANGE. FORCING FIT. BELT-POWER FORCING PRESS.

Digitized by Google

TABLE OF ALLOWANCES AND LIMITS OVER STANDARD FOR SHRINKING FITS.

Hole Diameter, Inches.	Shaft Diameter = Hole Diameter +	Limit Shaft Diameter + or −
.00 to .49	.00075	.00025
.50 to .99	.0015	.0005
1.00 to 1.49	.002	.0005
1.50 to 1.99	.0025	.0005
2.00 to 2.49	.003	.0005
2.50 to 3.24	.0035	.0005
3.25 to 3.99	.004	.0005
4.00 to 4.99	.0045	.0005
5.00 to 5.99	.0055	.0005
6.00 to 8.00	.0075	.0005

149. To shrink tires on wheel centers. — Wheel centers are turned standard and the allowance is made in boring the tire.

The allowance for shrinkage in rings or jackets for guns in the United States naval gun factories varies slightly for different classes of guns, but it is generally about .001″ to the inch. That is, if the diameter is 12″ the shrinkage will be .012″.

TABLE OF ALLOWANCES FOR LOCOMOTIVE DRIVING WHEEL TIRES FOR SHRINKING FITS.

Wheel Center Diameter, Inches.	Bore of Tire = Wheel Center Diameter −	Wheel Center Diameter, Inches.	Bore of Tire = Wheel Center Diameter −
38.00	.040	56.00	.060
44.00	.047	62.00	.066
50.00	.053	66.00	.070

150. Standard holes. — The holes referred to in tables of allowances, § 149, for the various kinds of fits are either bored or bored and reamed, and are within limits given in table of .00025″. They are tested by plug and limit gages.

Digitized by Google

TABLE OF LIMITS OF HOLE DIAMETERS ALLOWABLE UNDER AND OVER STANDARD.

Hole Diameter, Inches.	Limit Standard Diameter.	
	+	−
.00 to 1.24	.00025	.00025
1.25 to 2.49	.00075	.00025
2.50 to 5.99	.001	.0005
6.00 to 8.00	.001	.0075

151. Tables of allowances and limits for standard fits, represent common practice, but are not intended to conform to every case that may arise in fitting. In making allowances for any fit, certain conditions must always be considered. For example, the allowances for forcing fits are for cast-iron hubs twice the diameter of machine-steel shafts, and subject to modifications for different conditions, as the amount of metal surrounding the hole, the length of hole, and the elasticity of the metal.

152. To fit by trial and correction, and by allowance.—When only a few pieces have to be fitted, fit by trial. But for many pieces, fit according to allowance given in tables.

153. To turn and file fits, and to grind fits.—Work that was formerly turned and filed in the lathe is now roughed out with high-speed steel tools and finished in the grinding machine. This process produces work quicker, and truer, cylindrically. See § 379.

154. To produce standard fits. — Fit ordinary calipers to a plug gage, reference rod or disk which has the proper allowance. Turn and file or grind shaft to fit caliper. Another method is to use limit caliper gages directly on work. See Limit Gages, § 162.

155. To produce standard fits with micrometers. — Turn work direct to diameter plus double depth of tool marks; the allowance for filing with a fine feed and sharp tools is from .003″ to .004″.

Digitized by Google

STANDARD AND LIMIT GAGES.

156. Gages are instruments of reference for standardizing measurements and for determining dimensions exactly or within limits.

When a piece is machined nearly to size, a gage is invaluble for determining the exact dimension, the tightness or looseness of the fit giving an idea of its size.

157. Standard cylindrical gages, ring and plug. — Fig. 93 represent an accurate subdivision of the Imperial yard.

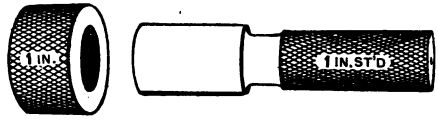


FIG. 93. — STANDARD RING AND PLUG GAGES.

These gages are made within various limits of accuracy, such as .0002″, .0001″, .00005″, .00002″.

158. Caliper gages, Fig. 94. — The measuring faces of the

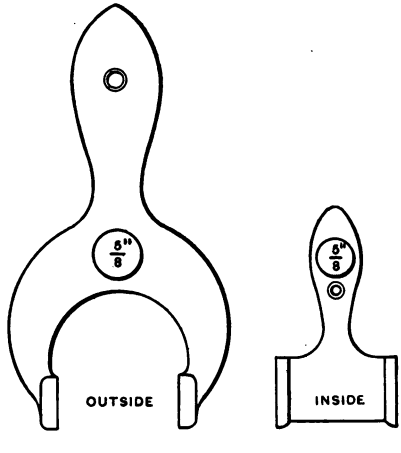


FIG. 94. — CALIPER GAGES.

outside gage are flat, the inside cylindrical. For general use, this gage is preferred to plug and ring gages.

Digitized by Google

159. Reference disk, Fig. 95, is used for testing and setting calipers.

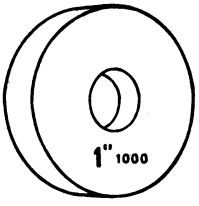


FIG. 95. — STANDARD REFERENCE DISK.

Attention. — As the heat from the hand will enlarge a gage perceptibly, the body of gages for very accurate testing is often covered with a jacket of rubber or wood, or provided with a hole to receive a wooden handle, as rubber and wood are poor conductors of heat.

160. End-measuring rod, Fig. 96, for gaging rings, cylinders, setting ordinary calipers, etc. The ends are sections of true spheres having diameters equal to the length of the rod.

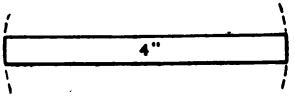


FIG. 96. — STANDARD END-MEASURING ROD.

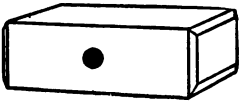


FIG. 97. — STANDARD END-MEASURE TEST PIECE.

161. End-measure test piece, Fig. 97, is used for testing micrometers, caliper gages and setting tools in planer and shaper. See *Advanced Machine Work.*

162. Limit gages are for use in the manufacture of duplicate pieces of work to save the time necessary in finishing to a single-gage dimension. They have two fixed dimensions which differ an amount equal to the limit allowed. See Tables,

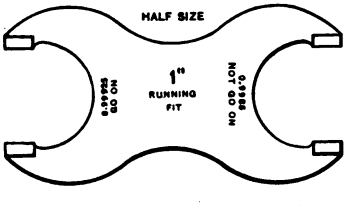


FIG. 98. — OUTSIDE LIMIT GAGE.

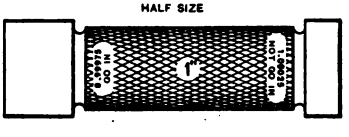


FIG. 99. — INSIDE LIMIT GAGE.

§§ 137, 138, 139, 140, 148, 149, 150. The gages, Figs. 98, 99, are clearly stamped with dimensions and directions.

Digitized by Google

Adjustable outside limit gages are obtainable which permit the limit to be varied to suit different classes of work.

163. Limit caliper gages for roughing out work are made from .003″ to .01″ larger than the maximum diameter of the finished piece, depending on the class of work.

164. Special gages. — Various other standard and special gages are obtainable.

Attention. — Care must be exercised to clean and oil gages before using and also not to force them into or over work with undue pressure or they will wear excessively and soon become unreliable.

Digitized by Google

CHAPTER VII.

TAPERS IN MACHINE CONSTRUCTION. TAPER TURNING AND FITTING. TAPER ATTACHMENT. STRAIGHT TURNING AND FITTING.

TAPERS IN MACHINE CONSTRUCTION.

165. Tapers are expressed as so much per unit of length, as 1″ per foot; that is, a piece 1′ in length would be 1″ larger at one end than at the other, as at *A* and *B*, Fig. 100. They

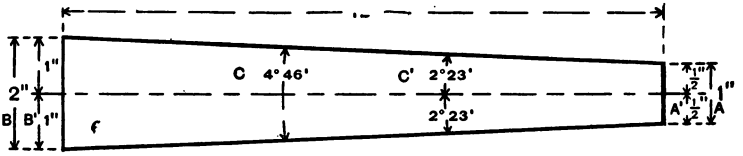


FIG. 100. — DIAGRAM FOR READING TAPERS.

may be expressed as so much per foot from the center line; as, ½″ per foot from center line, ½″ at *A′* and 1″ at *B′*, the same taper as 1″ per foot. They are also expressed in angular measurement, see Table of Tapers and Angles, *Advanced Machine Work*, as at *C*, angle of 4° 46′, or as at *C′*, 2° 23′.

166. Tapers, standard and special. — Taper parts are used on nearly all machines. The Morse taper system, approximately ⅝″ per foot (see §§ 648, 649), is used on drills and in drilling-machine and lathe spindles; the Brown & Sharpe taper system, ½″ per foot (see §§ 650, 651), in milling-machine spindles. Both are designated by name and numbers.

The Jarno taper system, .600″ per foot, is used to some extent in lathe and grinding-machine spindles. See §§652, 653.

Special tapers are used for spindle boxes, pins, and similar work.

Digitized by Google

167. Methods of turning taper. — Set over footstock, Fig. 101, or use a taper attachment, Figs. 105, 106. The first method can only be used for outside tapers; the others can be used for either outside or inside tapers.

168. To calculate distance to set over footstock. — Multiply one-half taper per foot (in inches) by whole length of work or mandrel (in feet).

Example. — The Morse Taper No. 3, Fig. 101, is .602″ per foot, length of work 8″. Find amount to set over footstock.

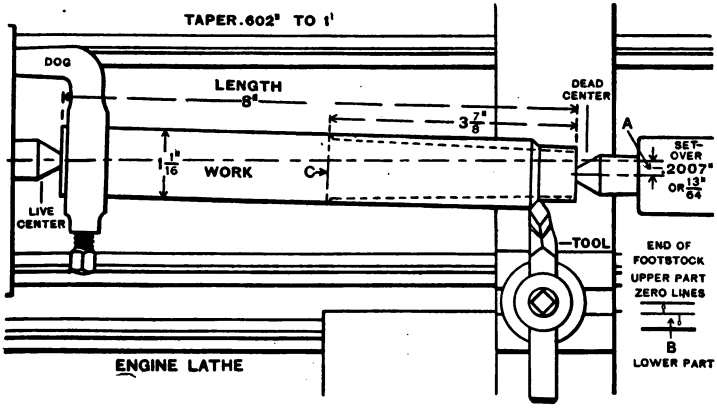


FIG. 101. — TURNING TAPER WITH FOOTSTOCK SET-OVER.

Solution. — $\frac{.602}{2} \times \frac{8}{12} = .2007''$ or $\frac{13}{64}''$.

The distance the centers enter the work affects this rule so slightly that it is ignored.

169. To set over footstock calculated distance, unclamp footstock and rotate adjusting screws to move dead center forward .2007″ or $\frac{13}{64}''$, as at *A*, Fig. 101, measuring lines *B* with dividers or rule set to .20″. See Table of Footstock Set-over for Morse, and Brown & Sharpe Tapers. See §§ 654, 655.

Digitized by Google

170. To calculate set-over when only length and diameters are given. Fig. 102.

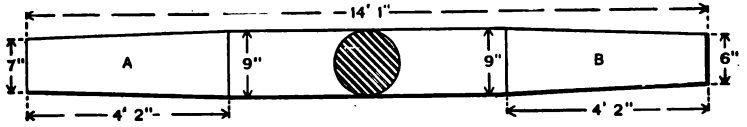


FIG. 102. — EXAMPLE IN LONG TAPER TURNING.

$$\textit{Formula.} — \frac{\text{Total length}}{\text{Length taper portion}} \times \frac{\text{Difference in diameters}}{2} = \text{Set-over.}$$

Example. — To find set-over; total length 14′ 1″; tapered portion 4′ 2″; difference in diameter 9″ − 6″ = 3″.

Solution. — 14′ 1″ = 169″; 4′ 2″ = 50″.

$\frac{169}{50} \times \frac{3}{2} = 5.07''$ (set-over).

171. To use a pattern to obtain set-over. — If a piece of taper work has to be duplicated, use it as a pattern to obtain approximate set-over by mounting it on centers and using a *test indicator*. See Tapering Pulleys, § 316.

TAPER TURNING AND FITTING.

172. To turn, file and fit Morse tapers No. 3 (.602″ to 1′), Figs. 103 and 104.

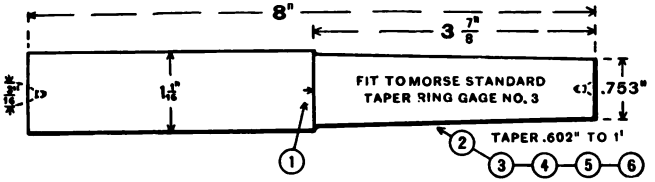


FIG. 103. — SCHEDULE DRAWING OF TAPER TURNING AND FITTING.

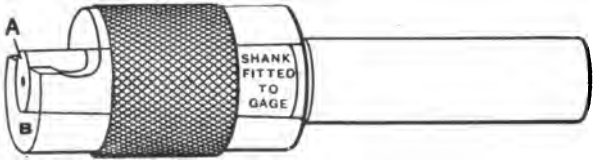


FIG. 104. — MORSE STANDARD TAPER RING GAGE, NO. 3.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES, AND TOOLS.

ROUGHING FINISHING

Stock, iron casting $\frac{1}{8}''$ large.
True live center. Set dead center in alinement.
Use carbon-steel cutting tools. See *Exception*, p. 59.

OPERATIONS.	MACHINES, SPEEDS, FEEDS.	TOOLS.
Snag, center, rough-square, recenter, finish square. See § 112.	Centering machine, Engine lathe, 12″ to 16″.	Vise, chisel, file, round-nose tool, side tool, 15° rake.
Straight turn. Rough one cut, finish one cut.	.....	Dog diamond-point tool, 15° rake, rule, calipers.
Lay off length of taper, (1).	.....	Vise, rule, scriber, chalk or copper sulphate.
Set over footstock to .2007″ or $\frac{13}{64}''$ as in Fig. 101 or set taper attachment to approximate taper. Set tool height of centers and rough turn $\frac{13}{16}''$ at small end, one cut, 2. When within $\frac{1}{16}''$ of *C*, Fig. 101, release power feed and use hand feed to within $\frac{1}{64}''$ of *C*.	2d speed, or 35 F.P.M. Power feed — 80 to 1″.	Diamond-point tool, 15° rake, calipers, rule.
Grind and oilstone tool. Take a trial cut about .004″ to .005″ (3). Piece will enter gage about $2\frac{1}{2}''$, enough to determine fit.	3d speed, or 50 F.P.M. Power feed — 140 to 1″.	Round-nose tool, Morse taper ring gage, No. 3, Fig. 103.
Clean hole and work. Chalk line along work and test in gage with rotating motion. If chalk line shows contact throughout by even rubbing, taper is correct. If only large end bears, set footstock backward slightly or vice versa, and take another trial cut. Repeat above until taper is correct.	.....	Round-nose tool, Morse taper ring gage, No. 3, chalk or Prussian blue.
Rough turn one cut to $\frac{49}{64}''$, 4, or until it will reach within $\frac{7}{32}$ of end of gage.	2d or 3d speeds, or 35 to 50 F.P.M. Medium power feed.	Diamond-point tool, 15° rake, calipers, rule.
Regrind and oilstone tool. Finish turn one cut to .753″ at end + .004″ for filing until end of work comes within $\frac{1}{16}''$ of end of gage (5).	3d speed, or 50 F.P.M. Fine power feed.	Round-nose tool, 1″ micrometer.
File lightly all over to remove tool marks, (6). Chalk line on work. Test work in gage, and file bright spots. Continue until fit is uniform and end *A* is even with end of gage, *B*, Fig. 104, or .753″.	Engine lathe, 4th speed, or speed lathe, 3d or 4th speed, or 105 F.P.M.	8″ or 10″ mill bastard file, file card, Morse taper ring gage, No. 3, 1″ micrometer, chalk or Prussian blue.

Attention. — Turning tools must be set at exact height of center. Take light finishing cuts. Avoid excessive filing. For very accurate work, coat walls of gage with Prussian blue to test taper.

Digitized by Google

TAPER ATTACHMENT.

173. To turn taper with taper attachment. Figs. 105, 106.—These attachments are applied to engine lathes to turn

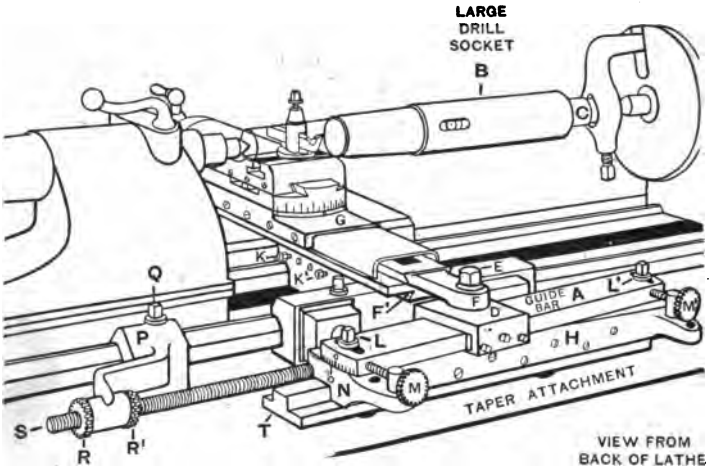


FIG. 105. — TURNING A TAPER WITH TAPER ATTACHMENT.

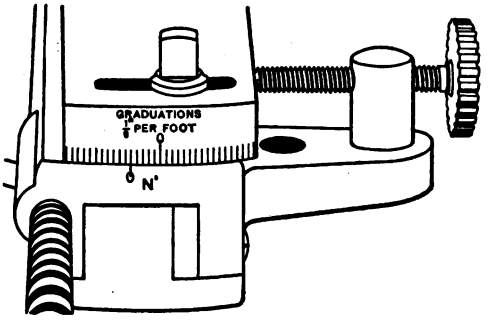


FIG. 106. — TAPER ATTACHMENT SET TO TURN A TAPER OF $\frac{1}{8}$" PER FOOT.

outside and inside tapers. The footstock does not have to be set over, and the length of the work does not have to be considered.

Digitized by Google

SCHEDULE OF OPERATIONS.

Figs. 105, 106.

Guide bar *A* is set to turn taper of drill socket *B*. Stub mandrel *C* provides a center for hollow end of socket. Guide bar *A* carries gibbed sliding block *D* which is connected by bolt *E* to extension *F* of supplementary slide *F'* which forms the base of cross slide *G*. Guide bar is swiveled upon its base *H* by unclamping screws *L*, *L'*. Drill socket *B* is to have a Morse taper, No. 4, .625″ or $\frac{5}{8}$″ to 1′.

To Set. — Rotate adjusting screws *M*, *M'* until the graduations on scale at *N* indicate the taper. See *N'*, Fig. 106.

Graduations on one end of bar give a taper of $\frac{1}{8}$″ to 1′, and on the other, $\frac{1}{10}$″ to 1′.

Fasten clamp *P* by bolt *Q* and clamp nuts *R*, *R'* on screw *S*. Sliding block *D* and slide *T* move with carriage. Guide bar *A* and base *H* remain stationary.

Attention. — Take trial cuts and test in taper ring gage or in spindle, and make corrections by adjusting guide bar. See Taper Fitting, §§ 165–172.

Caution. — Take up back lash before each cut by running carriage back at least $\frac{1}{2}$″ beyond end of work to avoid turning a portion at end straight.

Note. — To use lathe for straight work without disturbing taper setting, remove nut *R* and move clamp *P* away.

STRAIGHT TURNING AND FITTING.

174. To turn and file a 1″ straight running fit.

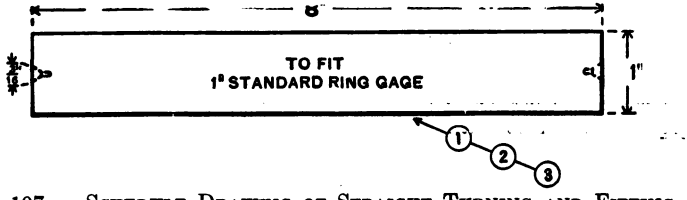


FIG. 107. — SCHEDULE DRAWING OF STRAIGHT TURNING AND FITTING.

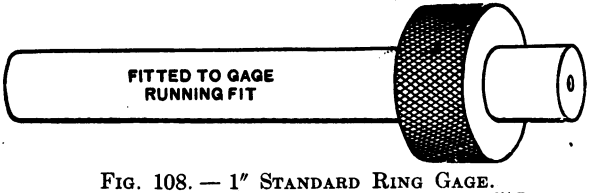


FIG. 108. — 1″ STANDARD RING GAGE.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES, AND TOOLS.

ROUGHING. FINISHING.

Stock, iron casting $\frac{1}{8}''$ large.

True live center. Set dead center in accurate alinement.

Use carbon-steel cutting tools. See *Exception*, p. 59.

OPERATIONS.	MACHINES, SPEEDS, FEEDS.	TOOLS.
TO MAKE FIT BY TRIAL AND CORRECTION.		
Snag center, rough square, recenter and finish square. See § 112.	Engine lathe 12″ to 16″.	Calipers, rule, round-nose tool, side tool, 15° rake.
Rough turn $1''+\frac{1}{64}''$, one cut, (1)....	1st speed, or 25 F.P.M. Power feed — 80 to 1″.	Copper under set screw of dog, diamond-point tool, 15° rake, calipers, rule.
Grind and oilstone tool. Oil mandrel and push in gage. Set calipers to large end close to gage. Take light cuts at end about $\frac{1}{8}''$ in length until calipers fit work slightly harder than on mandrel. Oil and try in gage, Fig. 108. When it enters about $\frac{1}{32}''$ with hand pressure, wipe off oil and take cut from end to end by reversing work, (2), one cut.	3d speed, or 50 F.P.M. Power feed — 140 to 1″.	Round-nose or diamond-point tool, 15° rake, calipers, 1″ mandrel, 1″ ring gage, oilstone.
File a small portion at end. Clean and oil work and gage. Try in gage. Wipe off oil after each trial. Continue fitting over whole length in this manner, (3).	Engine lathe, 4th speed, or speed lathe, 3d or 4th speed, or 110 F.P.M.	8″ or 10″ mill bastard file, file card, 1″ ring gage, oil.

Attention. — Test with calipers frequently. Avoid excessive filing. Test work in both ends of gage, as reamed holes are usually slightly larger at one end. The lack of oil when testing in gage will spoil both work and gage.

175. To make a 1″ running fit by allowance, Figs. 107 and 108.

Turn to	1.003″		
	1.004″		Micrometer calipers
	and		
File to	.9995″		
	.9990″		

Digitized by Google

CHAPTER VIII.

LATHE TOOLS FOR STEEL OR WROUGHT IRON. HOLDERS AND CUTTERS. CUTTING-OFF TOOLS. TURNING STEEL.

LATHE TOOLS FOR STEEL OR WROUGHT IRON.

176. More rake is used on these tools than on those used for cast iron.

177. Right-side tool. — The top face *A*, Fig. 109, is given a side rake of about 35° and the side clearance is 10°. A lubricant may be used, but good results are obtained in squaring and

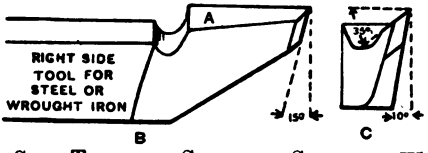


FIG. 109. — SIDE TOOL FOR SQUARING STEEL OR WROUGHT IRON.

turning small work dry. See Lubricants for Cutting Tools, § 100.

178. Step method of squaring. — To square a rough end or remove extra length, the step method is used. The side tool, Fig. 110, is fed inward about $\frac{1}{16}''$, then fed by hand long.

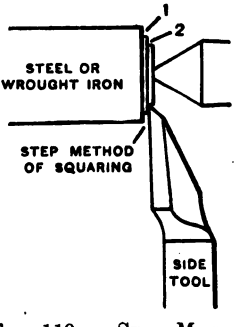


FIG. 110. — STEP METHOD OF SQUARING.

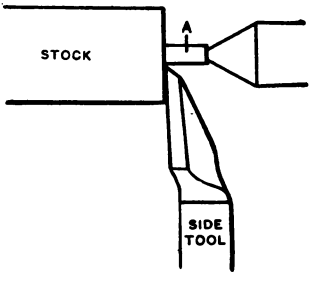


FIG. 111. — METHOD OF SQUARING LARGE AMOUNT OF STOCK.

Digitized by Google

feed. This cuts the first step, as at 1. The process is repeated at 2, and so on to the countersink. Then a continuous roughing cut is taken outward. When a forging or stock for a shaft, or spindle, is extra long, the first end is rough squared in the regular way, and the second end by the step method, as at *A*, Fig. 111. The stem left is chipped and filed off. The stock is recentered and finish squared. If extra length exceeds $\frac{3}{8}$″ a cutting-off tool may be used.

179. Right diamond-point tool. — For machine steel or wrought iron the top face *A*, Fig. 112, is given a combination of front and side rake of about 35° for roughing or finishing. For carbon steel (annealed) it is usually given less rake, 25° or 30°, and the cutting speed reduced.

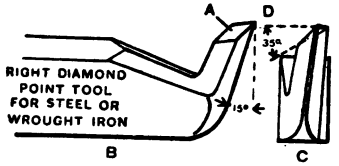


FIG. 112. — DIAMOND-POINT TOOL FOR TURNING STEEL OR WROUGHT IRON.

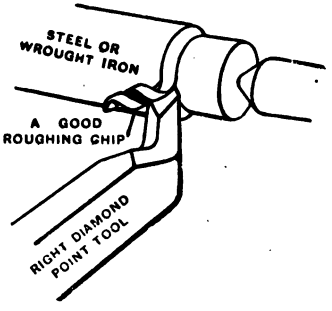


FIG. 113. — ROUGH TURNING STEEL OR WROUGHT IRON. GOOD.

180. To turn with a diamond-point tool. — Fig. 113 shows a diamond-point tool taking a roughing cut on machinery steel or wrought iron.

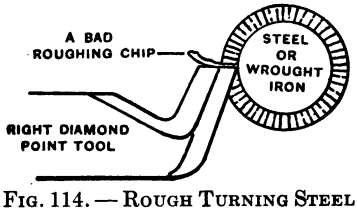


FIG. 114. — ROUGH TURNING STEEL OR WROUGHT IRON. BAD.

181. Steel or wrought-iron chips. — In Fig. 113 a chip cut by a tool with 35° rake, which produces a good surface, is shown, while in Fig. 114 the straight broken chip is cut by a tool without rake, which requires more power and leaves a ragged surface. Only the inexperienced would use a tool without rake for squaring and turning steel or wrought iron.

Digitized by Google

182. A small roughing tool, substitute for diamond-point, as in Fig. 115, is often used in roughing and finishing steel or

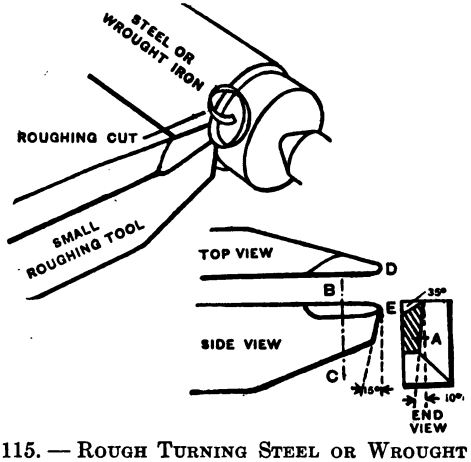


FIG. 115. — ROUGH TURNING STEEL OR WROUGHT IRON.

wrought iron. It may be used for heavy cuts on cast iron. It is given considerable side clearance, see end view, which shows at *A* a section taken at *BC*. The point is shaped as shown at *D* and *E*.

183. A large roughing tool for steel or wrought iron is shown in Fig. 116.

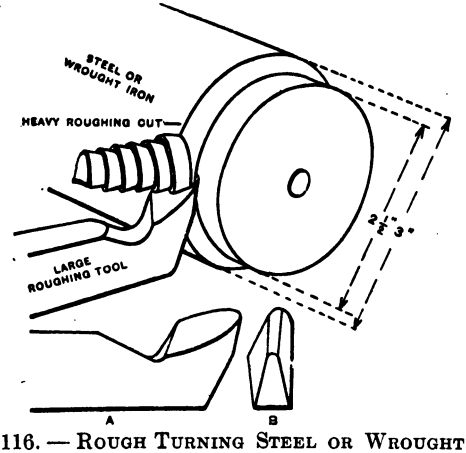


FIG. 116. — ROUGH TURNING STEEL OR WROUGHT IRON.

Digitized by Google

184. Large roughing tool ground from bar. — Fig. 117 shows a heavy roughing cut on a piece of tough nickel steel

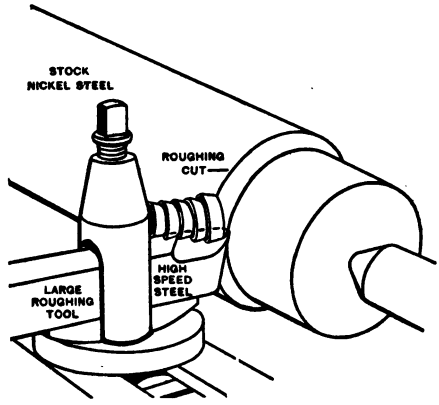


FIG. 117. — ROUGH TURNING STEEL OR WROUGHT IRON.

taken with a large roughing tool made of high-speed steel and ground from the bar.

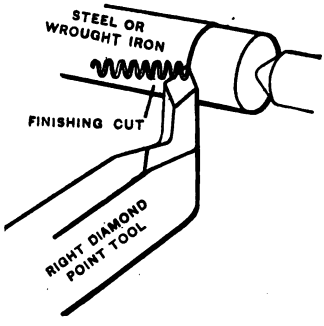


FIG. 118. — FINISHING STEEL OR WROUGHT IRON.

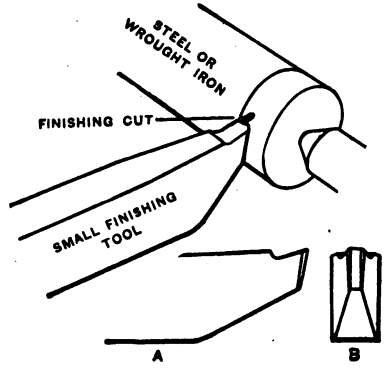


FIG. 119. — FINISHING STEEL OR WROUGHT IRON.

185. To finish turn steel or wrought iron.—The diamond-point tool in Fig. 118, with a fine feed, is best adapted to use

Digitized by Google

for finishing. To save time, small and large square-nose finishing tools, Figs. 119 and 120, are often used on large work with a

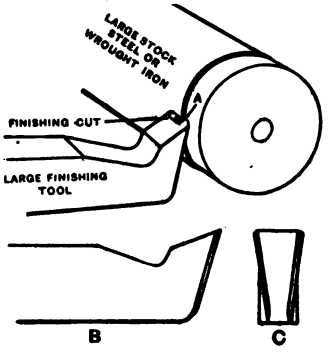


FIG. 120. — FINISHING STEEL OR WROUGHT IRON.

lubricant, light cut and coarse feed. The tools drag a little; that is, the back corners are set to cut deeper than the front corners to avoid chattering. See *A*, Fig. 123.

186. Spring finishing tool. — Fig. 121 shows a high-speed

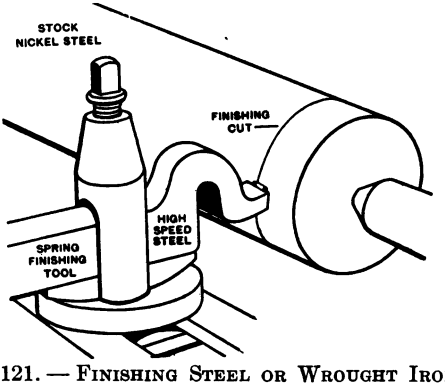


FIG. 121. — FINISHING STEEL OR WROUGHT IRON.

steel spring tool taking a light finishing cut with a lubricant on a shaft of nickel steel.

Attention. — Broad-nose tools have a great tendency to chatter and produce a rough corrugated surface caused by the long cutting edge, and made worse by frail, slender work, loose spindle bearings and loose cross slide.

Digitized by Google

187. Action of spring tools. — Broad-nose finishing tools have a tendency to dig into the work. The curved portion of tool in Fig. 121 serves as a spring, and when the cutting edge is set at the height of center, it will spring away from rather than into the work when a hard spot is encountered as the pivoting point is above the center. The success of spring tools depends upon having the proper amount of spring in proportion to length of cutting edge and diameter of work.

188. A shear tool, as in Fig. 122, is used, with a lubricant, to take a light finishing cut on steel or wrought iron.

Square-nose, spring and shear finishing tools should not be used until one has had considerable experience.

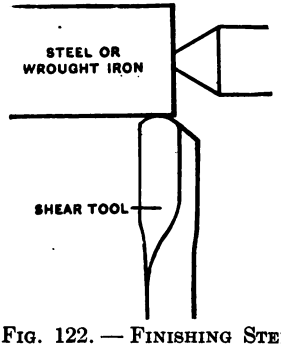


FIG. 122. — FINISHING STEEL OR WROUGHT IRON.

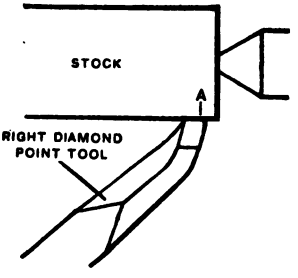


FIG. 123. — FINISHING STEEL OR WROUGHT IRON.

For long shafts that are rigid or well supported with a steady rest, a diamond-point, set as in Fig. 123, will produce an excellent finish either dry or with a lubricant. The cutting edge drags a little at *A*.

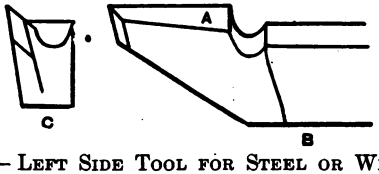


FIG. 124. — LEFT SIDE TOOL FOR STEEL OR WROUGHT IRON.

189. A left side tool is shown in Fig. 124. It is ground with the same angles as the right side tool and is used for squaring left shoulders.

Digitized by Google

190. A left diamond-point tool is shown in Fig. 125. It is used for turning from left to right.

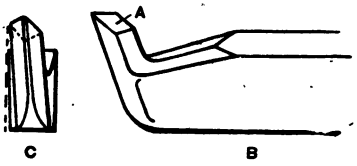


FIG. 125. — LEFT DIAMOND-POINT TOOL FOR STEEL OR WROUGHT IRON.

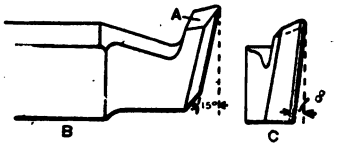


FIG. 126. — HALF DIAMOND-POINT TOOL FOR STEEL OR WROUGHT IRON.

191. A right half-diamond point tool is shown in Fig. 126. *A* is its top face. It can be used to turn up to and to square a shoulder.

HOLDERS AND CUTTERS.

192. Holders with inserted cutters, shown in chart, Fig. 131, are displacing forged tools to some extent. The cutters are of high-speed steel. The cutting angles are the same as on forged tools, and a student should become familiar with forged tools before using holders and cutters.

193. A straight holder, *A*, Fig. 127, supplied with a cutter *B*, held by screw *C*, is used as a substitute for a diamond-point tool.

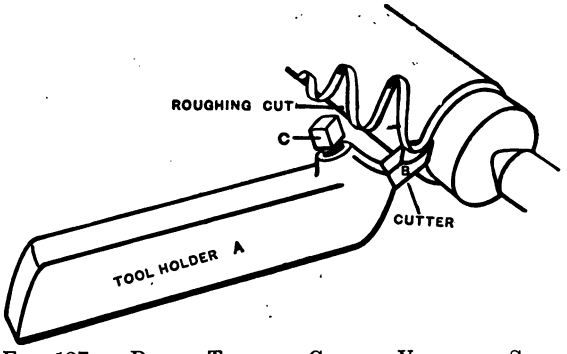


FIG. 127. — ROUGH TURNING CHROME-VANADIUM STEEL.

Fig. 129 shows at *D*, *E*, and *F* some uses to which a straight holder and cutters may be put; and Fig. 130 shows some uses of the right and left bent or offset holders and cutters *G* and *H*.

Digitized by Google

194. Useful forms of cutters. — For convenience, one should have a number of cutters ground to suitable cutting angles for different materials and for different operations, as *I*, *J*, *K*, *L*, *M*, *N*, Fig. 128.

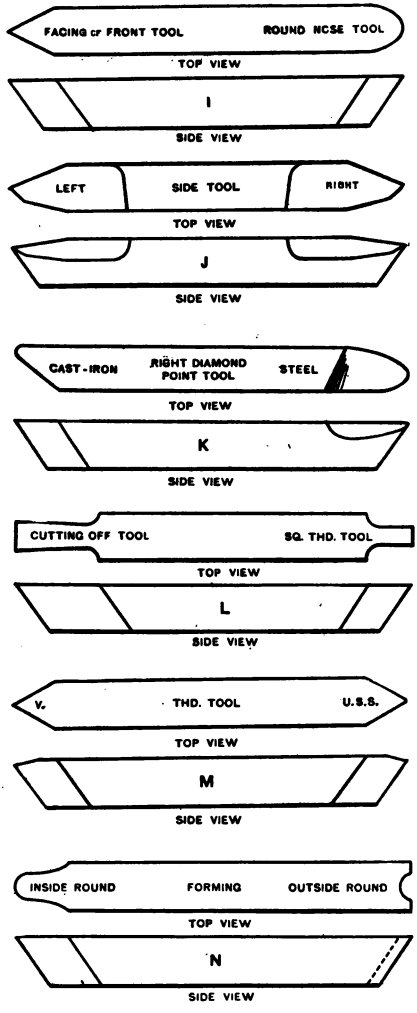


FIG. 128. — USEFUL FORMS OF HIGH-SPEED-STEEL CUTTERS.

Digitized by Google

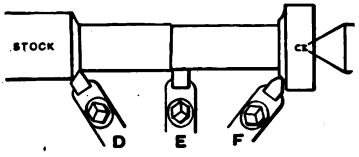


FIG. 129. — OPERATIONS WITH HOLDERS AND CUTTERS.

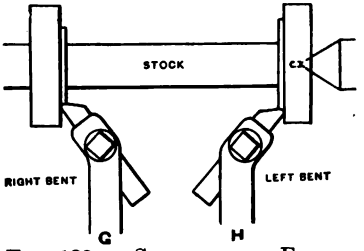


FIG. 130. — SQUARING OR FACING WITH OFFSET TOOLS.

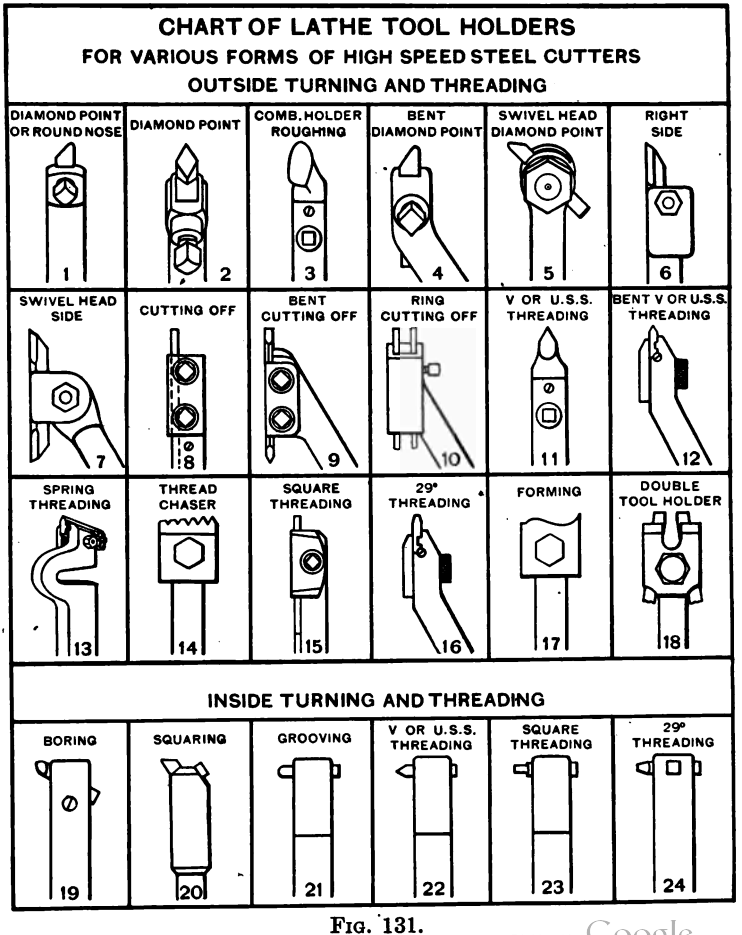


FIG. 131.

Digitized by Google

195. Double holder. — Fig. 132 shows a "home-made" double tool holder A, facing both sides of blank B in one

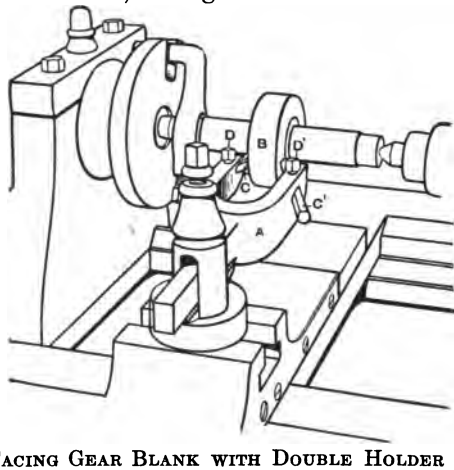


FIG. 132. — FACING GEAR BLANK WITH DOUBLE HOLDER AND CUTTERS.

operation. Cutters C, C' are adjusted to cut as desired, then clamped by screws D, D'.

For duplicate pieces, clamp lathe carriage and use index pointer on mandrel press to locate blank on mandrel.

196. Two forged tools for facing. — Two bent tools A

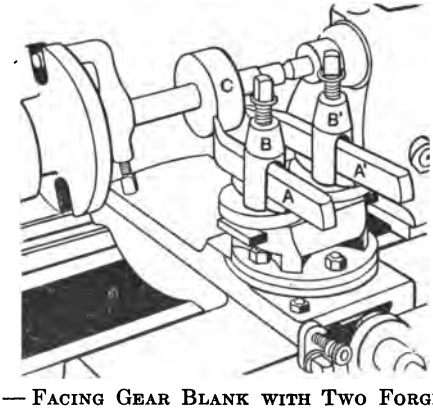


FIG. 133. — FACING GEAR BLANK WITH TWO FORGED TOOLS.

and A', Fig. 133, held in two tool-posts B and B' or by a bolt and strap, may be used to face the sides of gear blank C.

Digitized by Google

CUTTING-OFF TOOLS.

197. Cutting-off tool. — Fig. 134 shows a cutting-off tool for all metals. *A* is its top face. For clearance it is made

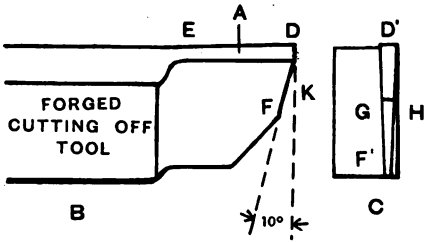


FIG. 134. — CUTTING-OFF TOOL FOR ALL METALS.

wider at the point, as shown. For steel or wrought iron a lubricant must be used, and the tool is often given front rake. For cast iron or brass the tool is used dry. The cutting speed is the same as for rough turning.

198. Cutting-off stock. — The tool is used close to the chuck jaws, as in Fig. 135, and should never be used to cut off stock

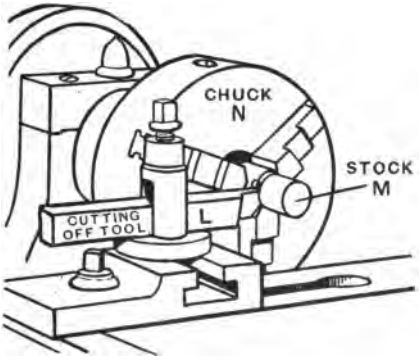


FIG. 135. — CUTTING-OFF STOCK IN ENGINE LATHE.

more than one diameter of stock away from chuck jaws lest it catch and break and also strain the chuck. Do not attempt to sever stock completely, as in Fig. 136, but finish cutting with a chisel or hack saw. Fig. 137 shows an offset cutting-off tool holder and cutter.

Digitized by Google

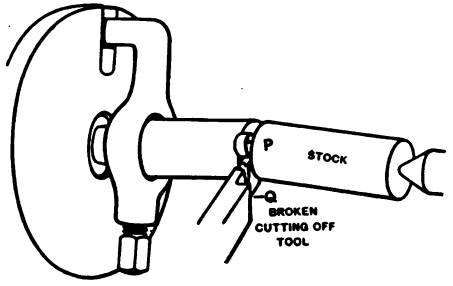


FIG. 136. — HOW TO BREAK A CUTTING-OFF TOOL.

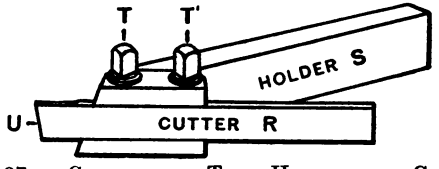


FIG. 137. — CUTTING-OFF TOOL HOLDER AND CUTTER.

TURNING STEEL.

199. To prepare two shaft blanks, one to turn and file running and driving fits (Fig. 138), and one to grind running and forcing fits (Fig. 139).

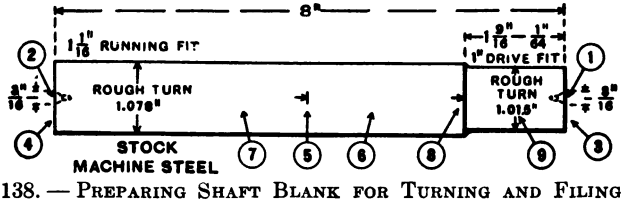


FIG. 138. — PREPARING SHAFT BLANK FOR TURNING AND FILING FITS. SCHEDULE DRAWING.

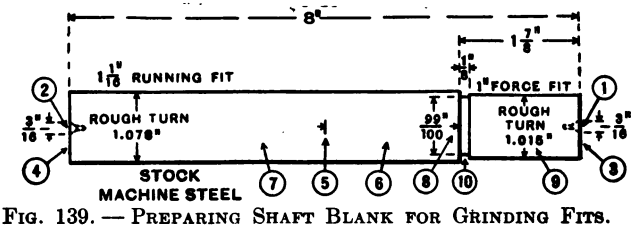


FIG. 139. — PREPARING SHAFT BLANK FOR GRINDING FITS. SCHEDULE DRAWING.

Digitized by Google

MULTIPLE SCHEDULE OF OPERATIONS, MACHINES, AND TOOLS.

ROUGHING.

Stock, machine steel $\frac{1}{16}''$ to $\frac{1}{8}''$ large. Machine dry, or use lard oil. True live center. Set dead center in approximate alinement. Use high-speed steel cutting tools. See *Exception*, p. 59.

OPERATIONS.	MACHINES, SPEEDS, FEEDS.	TOOLS.
Center (1), (2), Fig. 138 (1), (2), Fig. 139 to $\frac{3}{16}''$.	Centering machine. Drill speed, 1200 R.P.M. Countersink speed, 500 R.P.M.	$\frac{3}{32}''$ or No. 43 drill, 60° countersink, lard oil.
Rough square (3), (3), then (4), (4) to $8\frac{1}{64}''$.	Engine lathe 12″ to 16″. 2d or 3d speed, or 50 F.P.M. Hand or power feed.	Dog, side tool, 35° rake, or holder and cutter, calipers, rule.
Recenter to $\frac{3}{16}''$ and finish square (3), (3), then (4), (4).	3d or 4th speed, or 80 F.P.M. Hand or power feed.	
Mark lines approximately equidistant from ends, (5), (5). Rough turn to 1.078″ (6), (6). Reverse and turn (7), (7). One or two cuts.	1st or 2d speed, or 40 F.P.M. Medium power feed, —80 to 1″.	Two dogs with copper, diamond-point tool, 35° rake, or holder and cutter, micrometer.
Mark lines at (8), (8).		Copper sulphate, rule, scriber.
Rough turn (9), (9) to (8), (8). One cut.	2d speed, or 60 F.P.M.	
Cut recess to 0.99″ diameter, (10).	1st or 2d speed, or 30 F.P.M. Hand feed.	Cutting-off tool, lard oil, calipers, rule.
To grind fits, see § 419.		
To turn and file fits, see § 200.		

200. Turning and filing 1″ drive fit, (10), (11), Fig. 140.

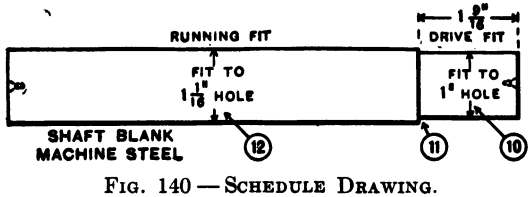


FIG. 140 — SCHEDULE DRAWING.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.

FINISHING.

Stock, shaft blank machine steel, rough turned 1.015″ diameter.

True live center. Set dead center in accurate alinement. Machine dry, or use lard oil.

Use high-speed steel cutting tools. See *Exception*, p. 59.

OPERATIONS.	MACHINES, SPEEDS, FEEDS.	TOOLS.
TO MAKE FIT BY TRIAL AND CORRECTION, (10).		
Push mandrel into reamed hole in work (flange or gear). Set calipers to large end close to work. Take several light cuts about $\frac{1}{8}$″ in length until calipers fit work slightly harder than on mandrel.	Engine lathe 12″ to 16″. 3d or 4th speed, or 70 F.P.M. Fine power feed, — 140 to 1″.	Dog, copper, 1″ mandrel, diamond-point tool, 35° rake, or holder and cutter, calipers, oilstone.
Hold work in vise. Oil shaft and press into large end of hole. When it enters from $\frac{1}{64}$″ to $\frac{1}{32}$″ with hand pressure, continue cut to shoulder.	Vise.	Copper jaws.
Square shoulder, (11).	3d or 4th speed, or 70 F.P.M. Hand feed.	Side tool, rule.
File small portion at end. Clean, oil, and press into hole. Continue filing sparingly and testing until shaft will enter hole one-third length of fit. Keyways are cut and key fitted before shaft is pressed or driven to shoulder.	Engine lathe, 4th speed, or speed lathe, 1st or 2d speed, or 175 F.P.M.	8″ or 10″ mill bastard, file, file card, oil.

Attention. — Hand pressure means grasping dog with both hands, and pressing shaft hard into hole with a right rotation. A left rotation will remove it. After each trial, the brightness of the surface and testing with calipers will indicate where to file. This is often called a wringing fit.

Digitized by Google

201. To make a 1″ drive fit by allowance, Fig. 140.

Turn to { 1.004″ ..... and 1.005″	.....	..... Micrometer. .....
File to { 1.00125″ ..... 1.00100″	.....	....
See Belt-power Forcing Press, Fig. 92.		

202. To turn and file $1\frac{1}{16}$″ running fit, (12), Fig. 140.

Shaft blank machine steel, rough-turned 1.078″ large.

Grind tools to 35° rake; machine dry, or use lard oil and for method of making fit, see § 174. For grinding fits, see § 419.

Digitized by Google

CHAPTER IX.

SCREW THREADS. THREADING TOOLS. THREADING OR SCREW CUTTING. BOLT AND NUT MAKING.

SCREW THREADS.

203. Forms of threads. — There are four common forms of threads: the Sharp V, Fig. 141; the United States Standard, Fig. 142; the Square, and the Acme Standard or 29° thread.

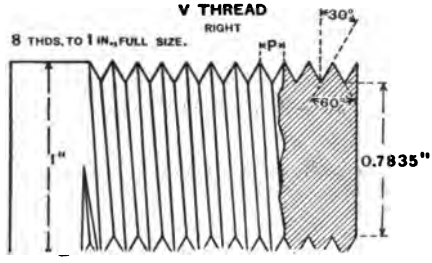


FIG. 141. — SECTIONAL VIEW OF SHARP V THREAD.

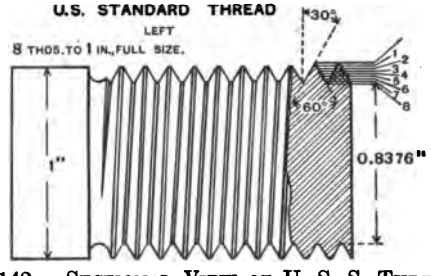


FIG. 142. —SECTIONAL VIEW OF U. S. S. THREAD.

The Whitworth (English) Standard thread, Fig. 161, is very little used in the United States. It is standard in Great Britain for coarse pitches. The British Association Standard thread is standard for fine pitches. It is similar in form to the Whitworth, but the angle is 47½ degrees. The International and French Standard threads, used with the Metric system in some foreign countries, are based on the same formulas as the United States Standard thread. See §§ 248, 636–638.

Digitized by Google

204. Right and left screw threads. — A right screw, Fig. 141, enters its nut when rotated to the right (clockwise). A left screw, Fig. 142, is the reverse. Screws are supposed to be right-threaded unless designated left.

205. Uses of different threads. — A right Sharp V or United States Standard thread is used to fasten parts together. A left thread is also used to fasten parts together, but only where a rotary motion would loosen a right thread as the nut on the near side of a wagon axle. The Square and 29° threads both right and left, are used to transmit motion.

206. Method of threading screws and nuts. — Screws for fine machine parts are threaded in a lathe. Bolts, studs, and screws are threaded with dies by power and by hand. See Dies, §§ 539–544.

207. Lead screws, taps, and worm screws may be milled with a thread milling machine.

208. Small nuts are threaded with a tap by hand or power

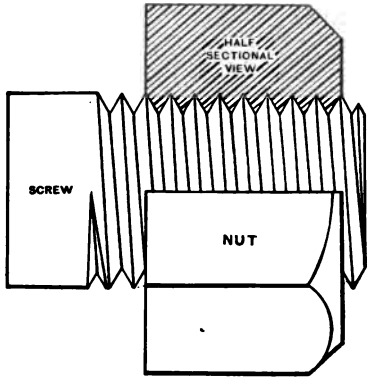


FIG. 143. — SCREW AND NUT.

(see Taps, § 510); large nuts are usually threaded in a lathe. Fig. 143 shows a screw and section of nut.

209. Rolled threads, produced by rolling the material between moving dies, are used on stove bolts, carriage bolts, some machine screws, etc.

210. Single and multiple threads. — Ordinary screws are single threaded, but for special purposes screws are double

Digitized by Google

threaded, triple threaded, etc. Examine the end of the screw; if only the end of one thread and one groove is found, it is single threaded, etc. See Multiple Threads, § 622.

211. The pitch of a thread is the distance along the axis of the work from the center of one thread to the center of the next, as *P*, Fig. 141.

212. The lead of a thread is the distance the screw advances in one revolution. In a single thread, the pitch is equal to the lead; in a double thread, the pitch is one-half the lead, etc.

213. Threads or pitches per inch signify the number of threads contained in one inch in length. For example, for $\frac{1}{10}''$ pitch we have 10 threads per 1″ (often called 10 P).

214. The diameter of threaded work is measured over the tops of the threads, as 1″, Fig. 141.

215. The root or bottom diameter is measured at the root or bottom of groove, as .7835″, Fig. 141.

216. Thread calipers, Fig. 144, may be used to test the diameter of Sharp V, United States Standard or 29° thread screws. As the fit is on the sides only the points are filed to fit the thread gage and are left truncated as at *A* and *A′* to avoid touching the bottom. The calipers are set to size by means of a thread gage, tap, or screw.

For accurate work, the screw is cut nearly to size of calipers and the final test is made by trying screw in the nut or work.

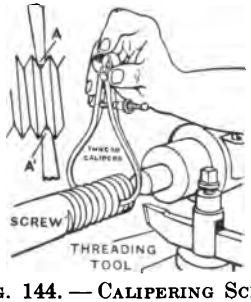


FIG. 144. — CALIPERING SCREW WITH THREAD CALIPERS.

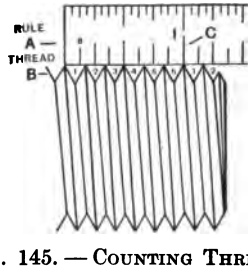


FIG. 145. — COUNTING THREADS WITH STEEL RULE.

217. To count threads. — Place rule *A* on thread *B*, Fig. 145, count grooves in one inch. To count Square and 29°

Digitized by Google

threads, place the end of rule against the right edge of one of the threads and count the spaces to the right. For double threads, triple threads, etc., count all the spaces and divide by 2, 3, etc., respectively.

A screw-pitch gage *A*, Fig. 146, is used to determine number of threads to one inch on a screw or in a nut. Handle *A*

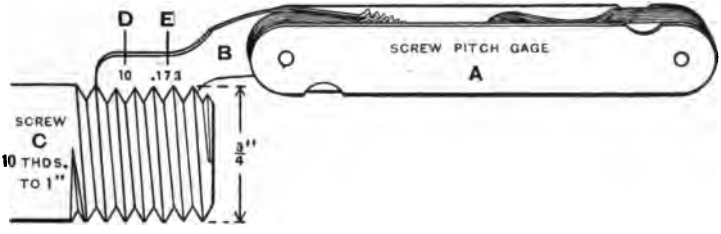


FIG. 146. — COUNTING THREADS WITH SCREW-PITCH GAGE.

contains many blades notched with all the ordinary threads to one inch. To determine number of threads to one inch, as at *C*, select by trial a blade that will match thread, and the number at *D* on blade, as 10, gives number of threads to one inch on screw. Decimal *E* is the double depth of a Sharp V thread of this pitch.

218. The Sharp V thread, Fig. 147. — The single depth for a 1″ pitch thread is .866″, double depth 1.732″. For pitch *P*, depth = .866 *P* = *D*. Double depth = 1.732 *P*.

$$P = \text{pitch} = \frac{1}{\text{No. of threads per inch}};$$

Root diameter = Outside diameter − double depth.

Formula: — Root diameter = Outside diameter − $\frac{1.732}{\text{No. of threads per inch}}$.

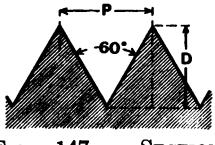


FIG. 147. — SECTION SHOWING PITCH AND DEPTH OF SHARP V THREAD.

Example. — To find root diameter of a screw 1″ diameter, 8 threads per 1″.

Solution. — $1 - \frac{1.732}{8} = .7835.$

Digitized by Google

219. Table of Sharp V-thread screws.

Diameter of Screw.	No. Threads per Inch.	Diameter of Screw.	No. Threads per Inch.	Diameter of Screw.	No. Threads per Inch.
$\frac{1}{4}$	20	$1\frac{3}{8}$	6	$3\frac{1}{2}$	$3\frac{1}{2}$
$\frac{5}{16}$	18	$1\frac{1}{2}$	6	$3\frac{3}{8}$	$3\frac{1}{4}$
$\frac{3}{8}$	16	$1\frac{5}{8}$	5	$3\frac{1}{2}$	$3\frac{1}{4}$
$\frac{7}{16}$	14	$1\frac{3}{4}$	5	$3\frac{5}{8}$	$3\frac{1}{4}$
$\frac{1}{2}$	12	$1\frac{7}{8}$	$4\frac{1}{2}$	$3\frac{3}{4}$	3
$\frac{9}{16}$	12	2	$4\frac{1}{2}$	$3\frac{7}{8}$	3
$\frac{5}{8}$	11	$2\frac{1}{8}$	$4\frac{1}{2}$	4	3
$\frac{11}{16}$	11	$2\frac{1}{4}$	$4\frac{1}{2}$	$4\frac{1}{4}$	$2\frac{7}{8}$
$\frac{3}{4}$	10	$2\frac{3}{8}$	$4\frac{1}{2}$	$4\frac{1}{2}$	$2\frac{3}{4}$
$\frac{13}{16}$	10	$2\frac{1}{2}$	4	$4\frac{3}{4}$	$2\frac{5}{8}$
$\frac{7}{8}$	9	$2\frac{5}{8}$	4	5	$2\frac{1}{2}$
$\frac{15}{16}$	9	$2\frac{3}{4}$	4	$5\frac{1}{4}$	$2\frac{1}{2}$
1	8	$2\frac{7}{8}$	4	$5\frac{1}{2}$	$2\frac{3}{8}$
$1\frac{1}{8}$	7	3	$3\frac{1}{2}$	$5\frac{3}{4}$	$2\frac{3}{8}$
$1\frac{1}{4}$	7	3	$3\frac{1}{2}$	6	$2\frac{1}{4}$

For Diameter of Tap Drills for Sharp V Threads, see § 538.

220. The United States Standard thread, Fig. 148, has its top and bottom truncated by $\frac{1}{8}$ of the depth, shown by divisions 1, 2, 3, etc., Fig. 142. The single depth of a 1″ pitch is .6495, double depth is 1.299″. For pitch P, depth = .6495 P = D. Double depth = 1.299 P.

$$P = \text{pitch} = \frac{1}{\text{No. of threads per inch}};$$

Root diameter = Outside diameter − double depth.

Formula. — Root diameter = Outside diameter −

$$\frac{1.299}{\text{No. of threads per inch}}.$$

$$F = \text{Flat} = \frac{P}{8}.$$

Digitized by Google

Example. — To find root diameter of a screw 1″ diameter, 8 threads per 1″.

Solution. — $1 - \frac{1.299}{8} = .8376''$.

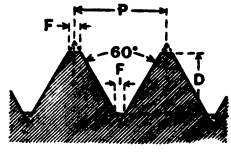


FIG. 148. — SECTION SHOWING PITCH AND DEPTH OF U. S. S. THREAD.

221. Table of United States Standard thread screws.

DIAMETER OF SCREW.	NO. THREADS PER INCH.	DIAMETER OF SCREW.	NO. THREADS PER INCH.	DIAMETER OF SCREW.	NO. THREADS PER INCH.
$\frac{1}{4}$	20	$1\frac{5}{8}$	$5\frac{1}{2}$	$3\frac{3}{8}$	$3\frac{1}{4}$
$\frac{5}{16}$	18	$1\frac{3}{4}$	5	$3\frac{1}{2}$	$3\frac{1}{4}$
$\frac{3}{8}$	16	$1\frac{7}{8}$	5	$3\frac{5}{8}$	$3\frac{1}{4}$
$\frac{7}{16}$	14	2	$4\frac{1}{2}$	$3\frac{3}{4}$	3
$\frac{1}{2}$	13	$2\frac{1}{8}$	$4\frac{1}{2}$	$3\frac{7}{8}$	3
$\frac{9}{16}$	12	$2\frac{1}{4}$	$4\frac{1}{2}$	4	3
$\frac{5}{8}$	11	$2\frac{3}{8}$	4	$4\frac{1}{4}$	$2\frac{7}{8}$
$\frac{3}{4}$	10	$2\frac{1}{2}$	4	$4\frac{1}{2}$	$2\frac{3}{4}$
$\frac{7}{8}$	9	$2\frac{5}{8}$	4	$4\frac{3}{4}$	$2\frac{5}{8}$
1	8	$2\frac{3}{4}$	4	5	$2\frac{1}{2}$
$1\frac{1}{8}$	7	$2\frac{7}{8}$	$3\frac{1}{2}$	$5\frac{1}{4}$	$2\frac{1}{2}$
$1\frac{1}{4}$	7	3	$3\frac{1}{2}$	$5\frac{1}{2}$	$2\frac{3}{8}$
$1\frac{3}{8}$	6	$3\frac{1}{8}$	$3\frac{1}{2}$	$5\frac{3}{4}$	$2\frac{3}{8}$
$1\frac{1}{2}$	6	$3\frac{1}{4}$	$3\frac{1}{2}$	6	$2\frac{1}{4}$

For Diameter of Tap Drills for U. S. S. Thread, see § 538.

THREADING TOOLS

222. The forged threading tool, Fig. 149, is forged, hardened, and then tempered to a light straw color. The clearance at *F D E* is 15°, and the cutting edges are *GH* and *JK*.

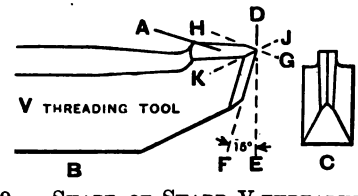


FIG. 149. — SHAPE OF SHARP V-THREADING TOOL.

Digitized by Google

The top is ground first and then the front faces are ground to fit center gage and the top is then set at height of center, as shown in Figs. 150 and 151 at *L*, *M*, *N* and *S*.

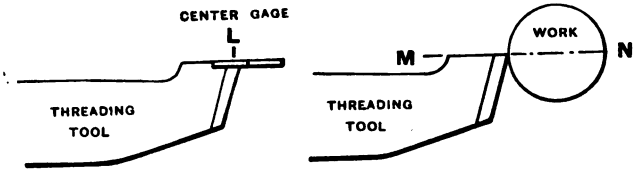


FIG. 150. — SETTING THREADING TOOL WITH CENTER GAGE AND TO HEIGHT OF CENTERS.

223. To set threading tool at height of center and at right angles to work. Fig. 151.

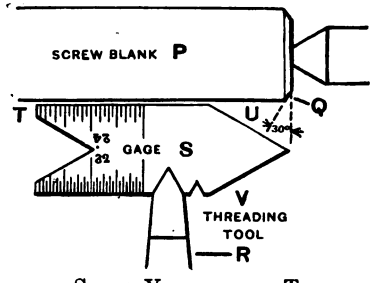


FIG. 151. — SETTING SHARP V-THREADING TOOL AT RIGHT ANGLES WITH WORK.

SCHEDULE OF OPERATIONS.

1. Turn work *P* to diameter.
2. Chamfer end *Q* depth of proposed thread.
3. Clamp tool *R* lightly with point 2″ from tool-post.
4. Adjust point to height of dead center (*MN*, Fig. 150).
5. Hold gage *S* as shown. Adjust cross feed.
6. Rap tool around until edge *TU* is parallel to work and about $\frac{1}{32}$″ from it.
7. Clamp tool firmly.

224. A special threading tool that may be used straight or bent to thread to a shoulder, is shown in Fig. 152.

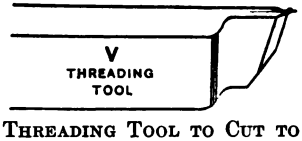


FIG. 152. — THREADING TOOL TO CUT TO SHOULDER.

Digitized by Google

225. United States Standard threading tool. — The United States Standard thread is cut with a V tool, *A*, Fig. 153,

FIG. 153. — SHAPE OF U. S. S. THREADING TOOL BEFORE TRUNCATING.

FIG. 154. — SHAPE OF U. S. S. THREADING TOOL.

truncated at point *B*, Fig. 154, $\frac{1}{8}$ depth of thread, which varies for every pitch. It is ground to fit notch *A* in U. S. S.

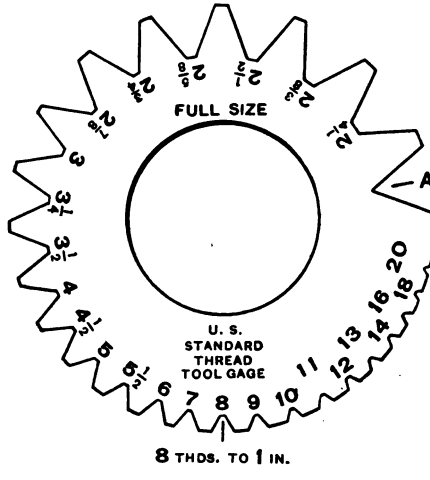


FIG. 155. — GAGE FOR U. S. S. THREAD TOOL.

thread gage, Fig. 155, then truncated at point to fit notch which corresponds with the threads to be cut, 8, Fig. 155. The tool is set the same as a Sharp V-thread tool.

Attention. — This thread is displacing the Sharp V thread.

Digitized by Google

THREADING OR SCREW CUTTING.

226. The theory of screw cutting in the engine lathe. — To thread a screw in a lathe, the threading tool is moved along the bed a positive and uniform amount for each revolution of the lathe spindle. This motion is obtained by means of a train of change gears which connect the lathe spindle to the lead screw, and by half nuts in the apron which connect the lead screw to the carriage.

On most lathes the first change gear is on a separate shaft called the stud. On some lathes this stud is geared to rotate at the same speed as lathe spindle, and on others at a different ratio.

Ordinary screws may be cut with simple gearing, — two change gears as in Fig. 156. See Compound Gearing, § 242.

227. To calculate simple gearing with spindle gear 1 to stud gear 1:

$$\frac{\text{Lead screw threads per inch} \times \text{constant}}{\text{Threads per inch to be cut} \times \text{constant}} = \frac{\text{teeth in gear on stud}}{\text{teeth in gear on lead screw}}.$$

The *constant* may be the common difference in number of teeth between the consecutive change gears, and this or any multiplier may be used to obtain available gears.

228. *Example.* — To cut 13 threads to 1″. Lead screw 5 threads to 1″; speed spindle same as stud; constant, 5.

Solution. —
$$\frac{5 \times 5}{13 \times 5} = \frac{25}{65} \begin{matrix} \text{(gear on stud.)} \\ \text{(gear on lead screw).} \end{matrix}$$

Attention. — For stud ratios other than 1 to 1, multiply threads per inch of lead screw by ratio of the stud gear to the spindle gear and proceed as before.

Digitized by Google

229. *Example.* — To cut 13 threads to 1″ spindle gear 1 to stud gear 2, Fig. 156. Lead screw 8 threads per inch; constant, 6; speed of stud is one-half speed of spindle.

Solution. — $\frac{8 \times 2 \times 6}{13 \times 6} = \frac{96}{78}$ (gear on stud.) (gear on lead screw.)

230. A practice screw blank and nut for threading or screwing cutter in an engine lathe, United States Standard or Sharp V thread, Fig. 156.

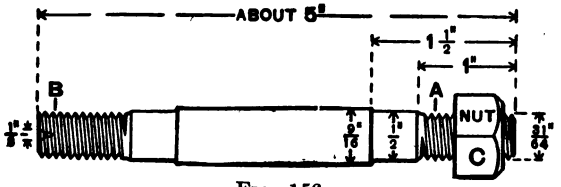


FIG. 156.

PREPARATION OF BLANK AND NUT.

Stock, machine steel or wrought iron. Rough and finish turn portions *A* and *B*, Fig. 155, to size for United States Standard thread, or $\frac{1}{64}$″ small for Sharp V thread. See Tapping and Threading Sizes, § 534. Tap nut *C* with ½″ United States Standard or Sharp V-thread tap. Use lard oil for tapping or threading.

True live center. Set dead center in accurate alinement.

231. Description of screw-cutting mechanism, Fig. 157.

A — Gear on spindle.	*L* — Stud.
B — Spindle.	*M* — Radial arm.
C — Gear driven by one or both idle gears.	*N* — Bolt to clamp *M* so that *F* drives through *KH* to *G*.
D & *D′* — Idlers for reversing.	*P* — Reversing lever; shifts bracket for cutting right or left threads.
E — Stud driven by *C*.	*Q* — Bracket carrying *D* and *D′*.
F — Stud gear.	
G — Lead screw.	
H — Lead-screw gear.	
K — Idler gear, loose on stud.	

Exception. — On lathes that do not have reversing gears *D* and *D′*, use two idlers to cut a left thread.

Digitized by Google

232. To set up lathe for threading or screw cutting. Fig. 157.

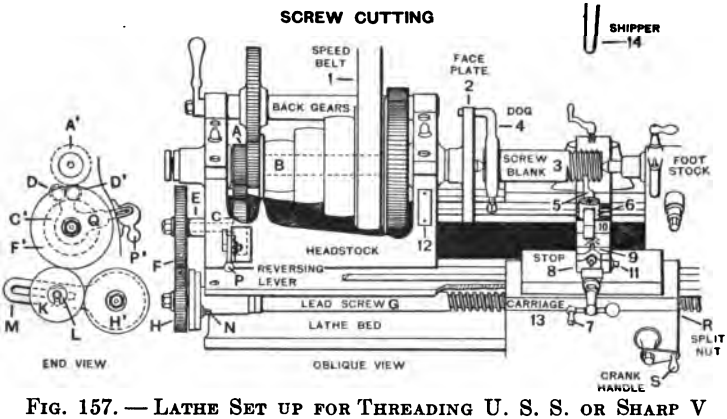


FIG. 157. — LATHE SET UP FOR THREADING U. S. S. OR SHARP V THREAD — SIMPLE GEARING, — TWO GEARS AND AN IDLER GEAR.

SCHEDULE OF OPERATIONS AND TOOLS.

CHANGE OF GEARS, THREADING TOOL, CENTER GAGE, OIL BOX.

I. Place belt on step 1, screw face plate 2 hard against shoulder.

II. On blank 3, fasten dog 4 and mark with chalk the place in face plate where dog is inserted. (Always return dog to marked slot or tool will not resume its cut.)

III. Grind tool 5 to form of thread desired, U. S. S. or Sharp V, fasten tool in post 6, and set with gage, Figs. 149 and 150.

IV. Feed tool in with handle 7 until it touches work, put on thread stop 8 with screw 9 in slide 10. Bring stop 8 against shoulder of 9 and clamp with screw 11.

V. Set footstock to allow $\frac{1}{2}''$ travel of tool beyond work.

VI. From index plate obtain gears for 13 threads and arrange as *FF'* and *HH'*.

VII. Select gear *K* for idler. Preferably a gear nearly the size of one of the change gears.

VIII. Place stud gear *F* on stud *E*, lead-screw gear *H* on *G*.

IX. Place gear *K* on stud *L*, in radial arm *M*, oil stud.

X. Adjust mesh of *H* and *K* and clamp in position.

XI. Swing radial arm *M*, to mesh *K* with *F* and clamp with bolt *N*.

XII. Be certain that friction feed is out before throwing in screw feed.

XIII. Connect lead screw to carriage 13 by handle *S* operating split nut *R*. Place tin box under tool to catch chips and drippings.

Digitized by Google

233. To operate the lathe to cut the thread.

SCHEDULE OF OPERATIONS.

I. Push shipper 14 toward headstock, adjust stop screw 9 and move tool 5 to trace a light line; stop lathe before tool reaches end of thread.

II. Finish length of thread by pulling belt by hand, the first time only.

III. Move tool out from work. Run carriage back. Adjust thread stop screw 9 to take cut.

IV. Count threads, Fig. 158. Terminate cut by power by easing tool out when about $\frac{1}{3}$ revolution from end of cut, or point of tool will snap off.

V. Stop lathe and reverse immediately.

VI. During return, lubricate work freely with lard oil and adjust stop screw 9 for next cut.

VII. Feed tool inward.

VIII. Start lathe forward and repeat.

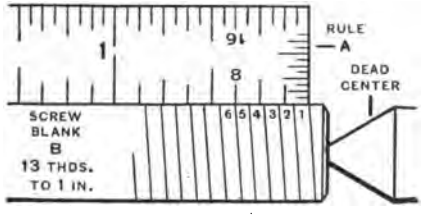


FIG. 158. — COUNTING THE TRACE OF A THREAD.

Attention. — The cutting speed for threading is about one-half to two-thirds that used for turning. The feed must be sufficient to allow tool to cut, for if tool is allowed to travel in groove without cutting, it will burnish and harden sides of thread, so that on next cut the tool will be likely to dig in and tear the thread.

Oilstone top face of tool just before taking last few light cuts.

234. Number of cuts for U. S. S. thread, 13 P. Take 8 cuts of .005″ each, 2 cuts of .002″ each, then 1 cut of .001″; clean, oil, and test in nut, and repeat until screw fits nut. Depth of thread when outside diameter is not reduced = .0499″.

235. Number of cuts for Sharp V thread, 13 P. Take 10 cuts .005″ each, 2 cuts .002″ each, then 1 cut of .001″; clean,

Digitized by Google

oil, and test in nut, and repeat until screw fits nut. Depth of thread when reduced $\frac{1}{64}''$ in diameter = .0588″.

236. To fit thread to nut. See § 216. Take light cuts as the thread approaches size, and after each cut clean and oil thread and try nut on. If a close fit is required, cut thread until nut will go on easily with a wrench. This smooths down the burr, after which the nut may go on with the fingers. For a hard fit, force nut on with a wrench. For a loose fit, cut the thread until the nut will go on with the fingers. After thread is fitted, chamfer the end to the depth and angle of the thread and file tops of thread slightly to remove burr.

237. To reset threading tool to resume cut. — If the tool is dull and thread is only partly cut, remove the tool, regrind and reset. If the end that receives the dog is cylindrical, loosen the dog, and rotate the work until tool fits the groove. Refasten the dog and feed tool away from work, run the lathe forward a few revolutions, by hand, to take up backlash; now notice if tool and groove match; if they do not, repeat operation. If end of work is square or hexagonal, as a bolt head, and driven by a clamp dog, disconnect lead screw from lathe spindle by reversing lever in headstock, or by removing stud gear, then adjust work.

238. To cut left threads. — Arrange gears as in cutting right threads, with the exception that the lead screw must rotate in opposite direction which is accomplished by the reversing gears *D*, *D′*, Fig. 157, or by an extra idler. At beginning of thread, cut a groove in which to start tool, and begin at left and cut to right.

239. To thread to shoulder or to terminate coarse thread. Cut groove or drill hole of diameter and depth equal to depth and width of thread. See Figs. 397, 408.

240. Fractional threads. — A fractional thread is one whose threads per inch are expressed by a mixed number, as $11\frac{1}{2}$ threads to 1″; or a fraction, as $\frac{3}{4}$ of a thread to 1″ ($1\frac{1}{3}''$ *P*).

To count fractional threads. — Take any number of threads

Digitized by Google

that will match with even inches on the rule, then divide the number of threads by the number of inches.

241. To calculate simple gearing to cut fractional threads. — Proceed as for whole threads.

Example. — To cut $11\frac{1}{2}$ threads per inch (1″ pipe tap). Lead screw 5 threads per inch; constant, 4; speed of stud same as speed of spindle.

$$\frac{5 \times 4}{11\frac{1}{2} \times 4} = \frac{20}{46} \begin{matrix} \text{(gear on stud.)} \\ \text{(gear on lead screw.)} \end{matrix}$$

242. Compound gearing. — To cut fine or coarse threads that are not obtainable with simple gearing, compound the gearing, using 4 gears, Fig. 159.

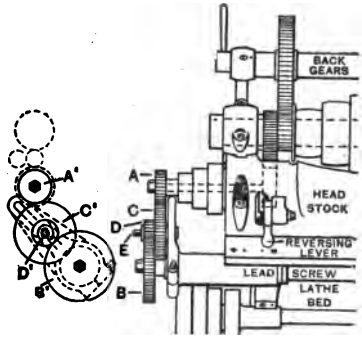


FIG. 159. — COMPOUND GEARING FOR THREADING, — FOUR GEARS.

To compound on some lathes, introduce between regular stud gear *A* and lead screw gear *B* two gears of different diameters, as *C* and *D*, which are feather keyed on a sleeve that runs freely on intermediate stud *E*, gear *A* driving gear *C*, and gear *D* driving gear *B*. The arrangement of gears is shown in end view at *A′*, *C′*, *D′*, and *B′*.

243. To calculate compound gearing. — Factor first term into two fractions and treat each separately.

Example. — To cut 60 threads per inch.

Lead screw 6 threads per inch; speed of stud same as speed

Digitized by Google

of spindle; constant, select multiple of 5, the common difference between gears.

Solution. — $\frac{6}{60} = \frac{2 \times 3}{5 \times 12}$. $\frac{2 \times 20}{5 \times 20} = \frac{40}{100}$. $\frac{3 \times 10}{12 \times 10} = \frac{30}{120}$.

Gear on stud A, 40; gear on lead screw B, 120; first gear on sleeve C, 100; second gear on sleeve D, 30.

Attention. — If more convenient, exchange drivers A and C or driven C and B.

244. To calculate compound gearing for fractional threads, proceed as for whole threads.

Example. — To cut $2\frac{1}{4}$ threads per inch $= \frac{2}{4}$.

Lead screw 2 threads per inch; speed of stud same as that of spindle; constant, multiple of 5.

Solution. — $\frac{2}{\frac{2}{4}} = \frac{8}{9} = \frac{2 \times 4}{3 \times 3}$.

$$\left.\begin{matrix} 2 \times 25 = 50 \\ 3 \times 25 = 75 \end{matrix}\right\} \text{one pair of gears.}$$

$$\left.\begin{matrix} 4 \times 15 = 60 \\ 3 \times 15 = 45 \end{matrix}\right\} \text{other pair of gears.}$$

Attention. — It often happens that one or more gears have to be made or bought. If the threads per inch are expressed decimally, as 2.833 threads per inch, proceed as before, selecting such multiple as will give available gears, using the nearest whole tooth in case of resulting fractional teeth.

245. Result of gearing calculations may be checked as follows:

For stud ratio one: Threads on screw to be cut $\times$ teeth of stud gear = threads on lead screw $\times$ teeth of lead-screw gear.

For stud ratios other than one and for compound gearing: Threads on screw to be cut $\times$ teeth of all drivers in succession = threads on lead screw $\times$ teeth of all followers in succession.

Digitized by Google

Example. — §228 $13 \times 25 = 5 \times 65 = 325$.
Example. — §229 $13 \times 1 \times 96 = 8 \times 2 \times 78 = 1248$.
Example. — §244 $60 \times 1 \times 40 \times 30 = 6 \times 1 \times 100 \times 120 = 72{,}000$.

246. To calculate gearing for a given lead. — First change to threads per inch by dividing one by the lead of screw to be cut, and proceed as before.

Example. — To cut screw $\frac{2}{3}''$ lead.

Lead screw 2 threads per inch; speed of spindle same as speed of stud; constant, multiple of 5.

Solution. —
$$\frac{1}{\frac{2}{3}} = \frac{3}{2} \text{ (threads per inch).}$$
$$\frac{2}{\frac{3}{2}} = \frac{4 \times 10}{3 \times 10} = \frac{40}{30}.$$

247. Compound gearing, ratio 2 to 1, is provided on some lathes on an extra adjustable stud. In such cases, gears are selected as in simple gearing for one-half or twice the desired pitch, and the 2 to 1 compound arranged to double the pitch or reduce it one-half.

248. To calculate gearing for metric screw threads with an English lead screw. — Gear up lathe as for cutting a Sharp V screw of the same number of threads per inch and use translating gears. One centimeter equals approximately $\frac{50}{127}$ of an inch. Provide lathe with pair of translating gears of 50 and 127 teeth. Arrange lathe as in compound gearing, Fig. 159.

Example. — To cut 13 threads to the centimeter.

Lead screw 5 threads per inch; speed spindle same as stud; constant, multiple of 5.

Solution. —
$$\frac{5 \times 5}{13 \times 5} = \frac{25}{65} \begin{matrix} \text{(gear on stud).} \\ \text{(gear on lead screw).} \end{matrix}$$

Place translating gears on feathered sleeve, meshing 50 with gear 65 and 127 with gear 25. A metric lead screw may be used for cutting English threads.

Digitized by Google

249. — "Catching the thread" (threading without a backing belt). — A method to save time in cutting long screws, by the quick return of the tool carriage after each cut by hand. If the thread is the same as lead screw or a multiple of it, throw half-nuts out at the end of each cut, return carriage by hand, throw in half-nuts, and the tool will resume its cut.

250. General method of "catching the thread." Fig. 160.

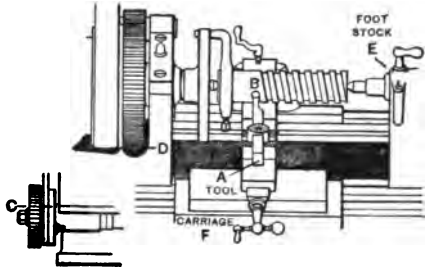


FIG. 160. — CATCHING THE THREAD.

SCHEDULE OF OPERATIONS.

I. Stop tool *A* at end of cut *B*.

II. Chalk tooth in lead screw gear which is opposite corner bed *C*.

III. Chalk tooth in headstock gear that coincides with front of headstock *D*.

IV. Withdraw tool and run lathe back by hand or power until tool passes end of work.

V. Run lathe forward to take up backlash and until chalked teeth *C* and *D* are in same position.

VI. Move footstock *E* to abut against carriage *F*, and clamp.

VII. Stop lathe each time tool is at end of thread, as at *B*.

VIII. Throw out half-nuts, return carriage by hand until it abuts against footstock, and throw in half-nuts.

Attention. — By watching chalk marks before throwing in half-nuts this method may be used without stopping lathe. Some lathes are provided with automatic thread stops. The tool cuts a groove at end of thread, after which lathe is stopped, half-nuts thrown out, and carriage moved along to a stop which is clamped on the bed.

Digitized by Google

251. Threading taper work. — Preferably use a taper attachment to cut a thread of correct pitch, as on pipe tap *A*, Fig. 161, as footstock "set over" will produce a thread slightly finer.

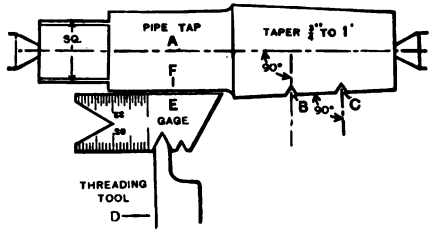


FIG. 161. — SETTING TOOL TO THREAD TAPER WORK.

Set thread tool 90° to the axis of work as at *B*, not as at *C*. Thread tool *D* with gage *E* against shank as at *F* is correct.

252. Whitworth (English) Standard thread. Figs. 162, 163. — The tops and the bottoms of the 55° threads are rounded, $\frac{1}{6}$ of pitch, as shown by divisions 1, 2, 3, etc.

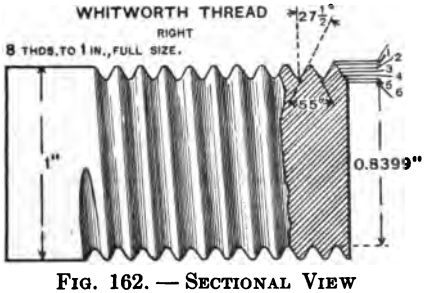


FIG. 162. — SECTIONAL VIEW OF WHITWORTH THREAD.

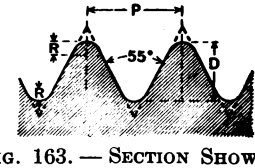


FIG. 163. — SECTION SHOWING PITCH AND DEPTH OF WHITWORTH THREAD.

The single depth of a 1″ pitch is equal to .64033, and the double depth 1.28066. For pitch *P*, depth = .64033 *P* = *D*. Double depth = 1.28066 *P* = 2 *D*.

$$P = \text{pitch} = \frac{1}{\text{No. of threads to 1 inch}}.$$

R = radius = .1373 *P*.

Root diameter = Outside diameter − double depth.

Digitized by Google

Formula. — Root diameter = Outside diameter — $\frac{1.28066}{\text{No. of threads to 1 inch}}$.

Example. — To find root diameter of a screw 1″ diameter, 8 threads to 1″.

Solution. — $1 - \frac{1.28066}{8} = .8399.$

253. Table of Whitworth (English) Standard threads.

Diameter of Screw.	No. Threads per Inch.	Diameter of Screw.	No. Threads per Inch.	Diameter of Screw.	No. Threads per Inch.
$\frac{1}{4}$	20	$1\frac{3}{8}$	6	$3\frac{1}{4}$	$3\frac{1}{4}$
$\frac{5}{16}$	18	$1\frac{1}{2}$	6	$3\frac{3}{8}$	$3\frac{1}{4}$
$\frac{3}{8}$	16	$1\frac{5}{8}$	5	$3\frac{1}{2}$	$3\frac{1}{4}$
$\frac{7}{16}$	14	$1\frac{3}{4}$	5	$3\frac{5}{8}$	$3\frac{1}{4}$
$\frac{1}{2}$	12	$1\frac{7}{8}$	$4\frac{1}{2}$	$3\frac{3}{4}$	3
$\frac{9}{16}$	12	2	$4\frac{1}{2}$	$3\frac{7}{8}$	3
$\frac{5}{8}$	11	$2\frac{1}{8}$	$4\frac{1}{2}$	4	3
$\frac{11}{16}$	11	$2\frac{1}{4}$	4	$4\frac{1}{4}$	$2\frac{7}{8}$
$\frac{3}{4}$	10	$2\frac{3}{8}$	4	$4\frac{1}{2}$	$2\frac{7}{8}$
$\frac{13}{16}$	10	$2\frac{1}{2}$	4	$4\frac{3}{4}$	$2\frac{3}{4}$
$\frac{7}{8}$	9	$2\frac{5}{8}$	4	5	$2\frac{3}{4}$
$\frac{15}{16}$	9	$2\frac{3}{4}$	$3\frac{1}{2}$	$5\frac{1}{4}$	$2\frac{5}{8}$
1	8	$2\frac{7}{8}$	$3\frac{1}{2}$	$5\frac{1}{2}$	$2\frac{5}{8}$
$1\frac{1}{8}$	7	3	$3\frac{1}{2}$	$5\frac{3}{4}$	$2\frac{1}{2}$
$1\frac{1}{4}$	7	$3\frac{1}{8}$	$3\frac{1}{2}$	6	$2\frac{1}{2}$

254. The Whitworth threading tool, Fig. 164. A tool of different size and shape is required for each pitch. It is made

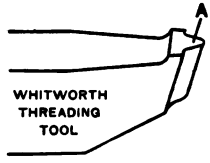


Fig. 164. — Shape of Whitworth Threading Tool.

similar to a formed cutter by milling. Grind the tool on top face *A*. Chasers and single point cutters and tool holders for Whitworth threads are obtainable.

Digitized by Google

BOLT AND NUT MAKING.

255. The bolt and nut-making operations that follow apply to all work of this class, as bolts, studs, nuts, and screws.

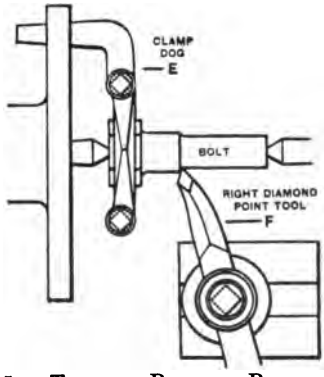


FIG. 165.—TURNING BODY OF BOLT TO HEAD.

To turn up to head of bolt, slant tool to left as at *F*, Fig. 165, and clamp tool firmly in tool-post.

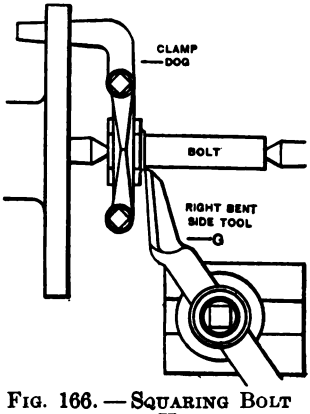


FIG. 166.—SQUARING BOLT UNDER HEAD.

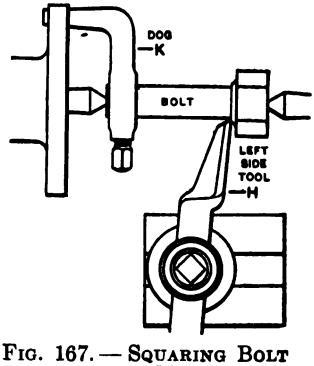


FIG. 167.—SQUARING BOLT UNDER HEAD.

To square under head, use right bent side tool *G*, Fig. 166, or left side tool *H*, Fig. 167. To drive from square or hexagonal heads use clamp or square dog *E*, Fig. 165, and to drive from stem use lathe dog *K*, Fig. 167.

Digitized by Google

256. Nut mandrels. — Nuts or similar pieces having tapped or threaded holes are screwed onto a threaded mandrel and rough and finish squared to thickness. In Fig. 168 nut *B* is screwed on to mandrel *A* against equalizing collar *C*, squared with side tool *D*, reversed and squared to thickness. It is then placed onto a milling mandrel and the flats milled in a milling machine. After this it is replaced on regular nut mandrel, and side tool *E* set at 45°, as near as can be determined by the eye, and the edge chamfered as at *F* to about

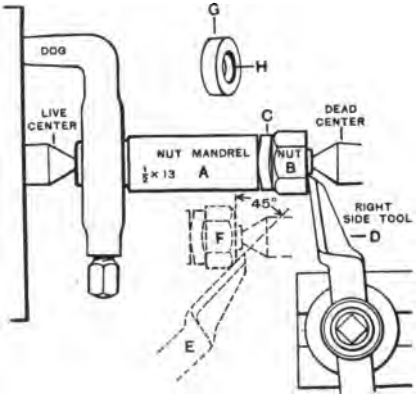


FIG. 168. — SQUARING AND CHAMFERING NUT.

$\frac{1}{32}$″ on edge of flats for ½″ nuts and more or less for larger and smaller sizes, to give them a neat appearance.

Instead of equalizing collar *C*, a plain collar *G* is often used, with one inside edge rounded to fit over filleted shoulder of mandrel. For ordinary work, nut mandrels are often recessed at shoulder and used without collars. Threaded mandrels, for work threaded while held in a chuck, as a face plate for a lathe or chuck, need no collar, as the work is faced true with hole while held in chuck, and this true face is screwed against the shoulder on the mandrel. See Mandrels, § 298.

Digitized by Google

257. Chamfering bolt head, nut, and screws. A clamp nut (spring or split nut), Fig. 169, is used to protect the thread and prevent dog from bruising it.

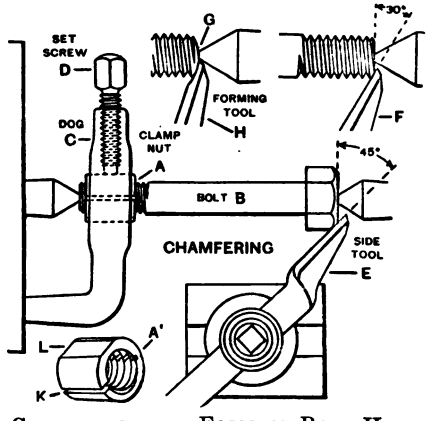


FIG. 169. — CHAMFERING AND FORMING BOLT HEAD AND POINT.

SCHEDULE OF OPERATIONS AND TOOLS.

Bolt Heads. 1. Place clamp nut *A* on bolt *B* and fasten dog *C* with set screw *D*. 2. Set side tool *E* at angle of 45°, estimated. 3. Chamfer bolt head to remove corners and give neat appearance. *Nuts.* Mount nut on mandrel and	chamfer same as bolts. See *E*, *F*, Fig. 168. *Screws.* Set side tool *F* at angle of 30° and chamfer to depth of thread. To round end of screw *G*, use side tool *H* shaped into a forming tool.

258. To make a clamp nut, A′, Fig. 169.

1. Drill, tap, square and turn piece of round carbon steel to size. 2. Slit at *K* and mill or file flat *L* to receive set screw of dog.	3. Harden and temper to a spring temper.

Attention. — To make an improvised clamp nut, slit a nut at a corner with a hack saw.

A clamp nut and dog may be used to set or remove studs.

Digitized by Google

259. To make a finished bolt $\frac{1}{2}$″ diameter. Fig. 170.

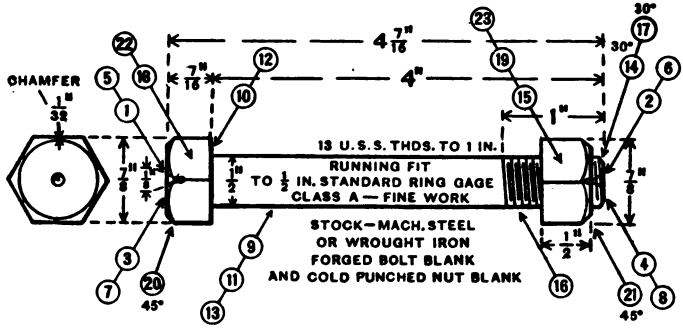


FIG. 170. — SCHEDULE DRAWING OF A BOLT.

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.

ROUGHING. FINISHING.

Stock, machine-steel or wrought-iron forging, $\frac{1}{16}$″ large.

Machine-steel or wrought-iron hexagonal nut blank $\frac{1}{16}$″ large.

True live center, set dead center in accurate alinement.

Lard oil may or may not be used in squaring and turning steel or wrought iron.

Use carbon-steel cutting tools. See *Exception*, p. 59.

OPERATIONS.	MACHINES, SPEEDS, FEEDS.	TOOLS.
Straighten and file ends flat.	Vise. Straightening press.	File, chalk.
Center, machine method, $\frac{1}{8}$″, (1,) (2), straighten.	Centering machine drill, 1700 R.P.M.; countersink, 600 R.P.M.; Straightening press.	$\frac{1}{16}$″ drill, 60° countersink, lard oil.
Mount on centers. Rough square to $4\frac{7}{16}$″ + $\frac{1}{64}$″, (3), (4). Take as little as possible off, (3). See step method of squaring, § 178.	Engine lathe 12″ to 16″. 3d speed, or 30 F.P.M. Hand feed.	Regular and clamp dogs, side tool, 35° rake, calipers, rule.
Recenter to $\frac{1}{8}$″, (5), (6).	Speed lathe, drill 4th speed, countersink 3d speed.	
Regrind and oilstone tool and finish square to $4\frac{7}{16}$″, (7), (8).	3d speed, or 50 F.P.M.	Side tool, 35° rake, rule.

Digitized by Google

Operations.	Machines, Speeds, Feeds.	Tools.
Rough turn to $\frac{1}{2}'' + \frac{1}{64}''$ (9) one or two cuts.	3d speed or 30 F.P.M. Medium power feed — 80 to 1″.	Clamp dog, diamond-point tool, 35° rake, calipers, rule.
Rough square under head, (10).	3d speed, or 30 F.P.M. Hand feed.	Left side or bent right side tool, calipers, rule.
Regrind and oilstone tool, and finish turn to fit gage with allowance for filing, using calipers, mandrel and gage to obtain size; or measure with 1″ micrometer and allow .003″ for filing. (11). One cut.	3d or 4th speed, or 50 F.P.M. Fine power feed — 140 to 1″.	Diamond-point tool, 35° rake, calipers, $\frac{1}{2}''$ mandrel, or 1″ micrometer, $\frac{1}{2}''$ ring gage.
Regrind and oilstone tool and finish square under head, (12).	3d speed, or 50 F.P.M. Hand feed.	Left side or bent right side tool, 35° rake, calipers, rule.
File to fit gage, running fit. Oil work with machine or lard oil when testing in gage. (13.)	Engine lathe, 4th speed, or speed lathe 8″ to 12″, 3d or 4th speed, or 175 F.P.M.	8″ or 10″ mill bastard file, calipers, $\frac{1}{2}''$ standard ring gage, oil, 1″ micrometer.
Chamfer point to 30°, (14).	Engine lathe, 3d speed, or 30 F.P.M. Hand feed.	Side tool, 35° rake.
Tap $\frac{1}{2}''$ nut blank by hand, square to thickness, (15).	Vise. 3d speed, or 50 F.P.M. Hand feed.	$\frac{1}{2}'' \times 13$ U.S.S. tap, oil, nut mandrel, calipers, rule.
Grind threading tool to fit gage, and thread bolt to fit nut. Make close fit, see Screw Cutting, § 226, rechamfer point, (16), (17).	Engine lathe, 1st speed, or 20 F.P.M. Arrange for 13 threads.	Clamp dog, 13 pitch U.S.S. thread tool, center gage, thread calipers, rule, oil.
Mill head and nut to size $\frac{1}{2}'' +$ .003″ for filing and polishing, (18), (19).	Milling machine. 3d or 4th speed. Back gears in, or 50 F.P.M. Medium power feed.	Heading mills, index head and chuck, milling machine nut mandrel, oil, 1″ micrometer.
Chamfer head and nut to 45°, (20), (21), §§ 256, 257.	Engine lathe, 3d speed, or 50 F.P.M.	$\frac{1}{2}'' \times 13$ clamp nut, and nut mandrel, rule, side tool.
File and polish flats. See *Elements of Machine Work*. Do not round corners, (22), (23).	Vise.	8″ or 10″ hand-smooth file, nut mandrel, copper jaws, 90 emery cloth, lard oil.

Digitized by Google

CHAPTER X.

CHUCKS. FACE PLATES. CHUCKING. REAMING.

CHUCKS.

260. The term chuck has a double meaning. First, it is the device used for holding work, drills or other tools. Second, it is the act of securing work in a holding device.

Chucks are indispensable to a large class of work.

261. Attaching chucks to machine spindles. — Drill and other small chucks are attached by a shank, one end fitting the chuck and the other the hole in spindle. Lathe chucks are usually attached by a threaded backing plate.

262. Chuck jaws. — Three general kinds, Fig. 171:

Drill jaws, for holding drills, rods, and similar pieces, also for holding hollow work by the inside.

Lathe jaws, for lathe work of large diameter.

Milling-machine jaws, used on milling machines.

Reversible jaws. — The jaws of some chucks may be reversed and used either as drill or lathe jaws.

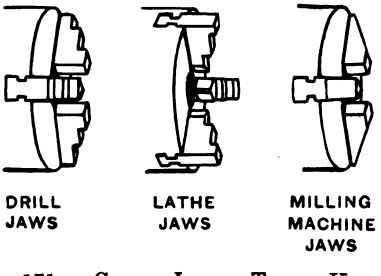


FIG. 171. — CHUCK JAWS. THREE KINDS.

263. Classes of chucks. — Drill, independent, universal and combination, besides draw-in chucks.

Digitized by Google

264. A drill chuck is used to hold drills and small work.

265. An independent chuck, Fig. 172, is one in which each jaw is moved independently with wrench. Chuck *A* consists of disk *B* screwed to spindle of headstock *C*. Lathe jaws *D*, stepped to suit different diameters of work, slide in slots in disk *B* and are moved by screws *E*, *E*, operated by a special wrench. Concentric circles are marked on the face of some chucks to facilitate setting work.

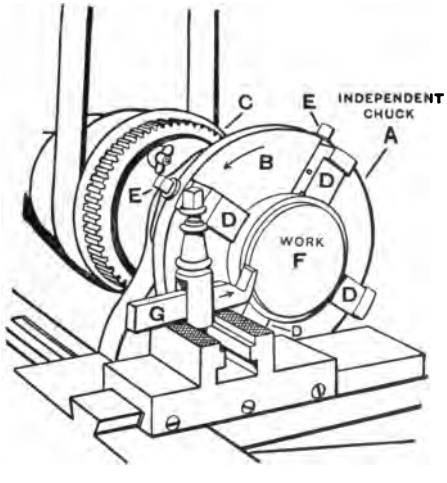


FIG. 172. — INDEPENDENT CHUCK. FACING DISK.

Attention. — Independent chucks are better adapted for rough work than universal chucks.

266. To true up and hold work in an independent chuck. — Grip work tightly. Run lathe at a moderately high speed, rest hand on carriage and hold a piece of chalk to just touch work. Stop lathe, loosen jaw or jaws opposite part marked by chalk and set others in. Erase chalk mark and test again, continuing until work runs true, then set all jaws up hard. Fig. 172 shows also the operation of facing work *F* with cutting-in tool *G*.

Digitized by Google

267. A universal chuck, Fig. 173, is one whose jaws move to and from the center simultaneously and concentrically. Chuck *A* is screwed to spindle of headstock *B*. This is known as a geared scroll chuck, and is made with either drill jaws, as shown, or lathe jaws. It consists of shell *C*, three bevel pinions *E* in mesh with an annular bevel gear, upon whose face is a scroll which engages jaws *D*. This chuck should be used for smooth work.

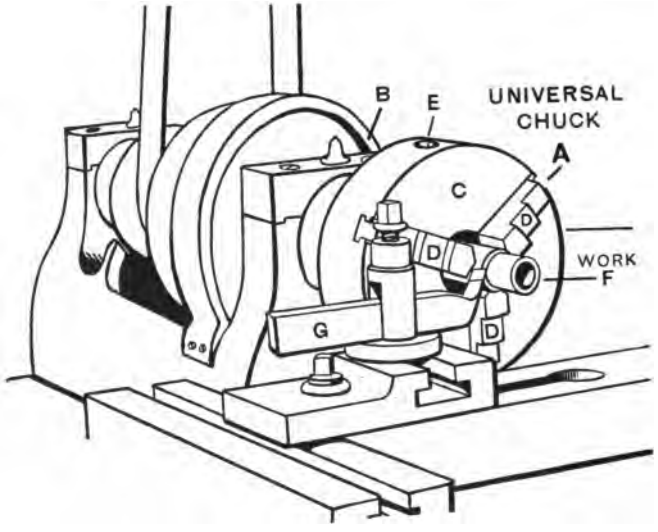


FIG. 173. — UNIVERSAL CHUCK. TURNING SLEEVE.

268. To true up and hold work in a universal chuck. — Place work in chuck, "set up" jaws by one pinion, run lathe and use chalk as before. If not true enough, loosen and turn work about one-quarter of a revolution; tighten pinion and test again; when right, tighten pinions hard.

Fig. 173 also shows how a bushing is made from bar *F*. The bar is squared, then drilled and reamed by chucking method (see §§ 281, 294), turned with tool *G*, and cut off with cutting-off tool.

Digitized by Google

269. A combination chuck, Fig. 174, is a chuck in which the jaws may be moved independently or simultaneously. When moved simultaneously, jaws may be set either concentrically or eccentrically. Chuck *A* consists of shell *B* and jaws *C* moved by screws *D*. These screws mesh with thread on back of jaws, and carry pinions which can be placed in mesh with an annular bevel gear controlled by device on back of chuck. When in mesh, chuck is universal; out of mesh, each jaw can be moved independently. In Fig. 174

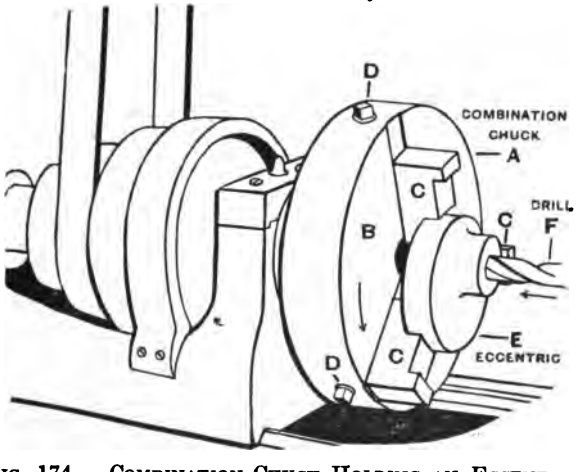


FIG. 174. — COMBINATION CHUCK HOLDING AN ECCENTRIC.

the jaws are set eccentric with annular out of mesh, then annular is thrown in mesh and the jaws are controlled as in a universal chuck. To make the chuck universally concentric, adjust each jaw to a circle on face of chuck and throw in annular.

Attention. — Independent and combination chucks usually have a number of concentric circles cut on their faces to facilitate setting the jaws true.

270. Special chucks can be made or ordered from a manufacturer. For some classes of work, jaws of special shape may be home-made to fit a regular chuck.

Digitized by Google

271. Face plate jaws are obtainable which may be bolted to a face plate and used as a chuck.

272. Draw-in (spring) chucks, or collets, Fig. 175, are used on toolmakers' and watchmakers' lathes and also on some engine and turret lathes to hold bars or rods, as *BB'*. The rod is passed through the spindle and accurately held by the chuck, and from the rod small screws, studs, bolts, etc., may be conveniently made without preliminary cutting off, centering, squaring, etc.

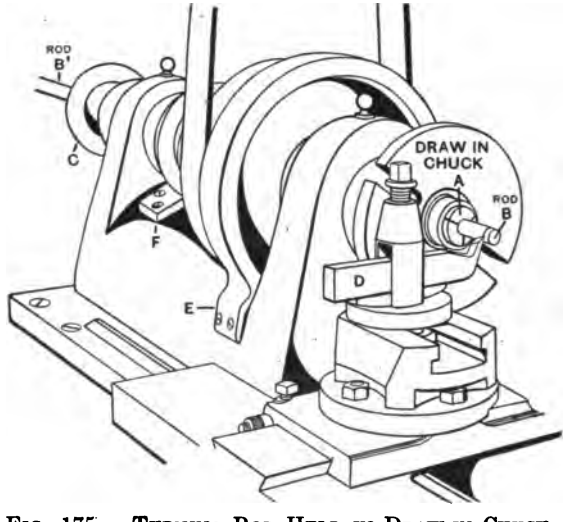


FIG. 175. — TURNING ROD HELD IN DRAW-IN CHUCK.

The steep taper on the chuck fits the conical hole in end of spindle. The chuck is slitted a short distance at three equidistant points, and rotating wheel *C*, which operates a hollow shaft that passes through the spindle and makes a threaded connection to end of chuck, draws the chuck into its seat to grip the bar or rod.

273. Care of chucks. — All chucks, and especially universal chucks, are short-lived for accurate work unless used intelligently and properly cared for; they should be cleaned and oiled frequently.

Digitized by Google

274. Mounting and removing chucks. — Clean and oil thread of both chuck and spindle, remove live center and plug hole with clean waste. Hold chuck square against nose of spindle with right hand or arm and rotate lathe with left hand until the chuck comes against shoulder on spindle. Small chucks may be loosened by grasping one jaw with a monkey wrench and striking the handle a sharp blow with the hand; large chucks, by placing a block of wood between a jaw and the bed of lathe and rotating lathe backward (with back gears in) by hand. Arbor or shank chucks are inserted and removed the same as lathe centers.

Attention. — To avoid springing work held in a chuck, the jaws should be forced against the solid parts, if convenient, as the arms of a pulley.

In some classes of light work, it is often necessary to loosen the jaws slightly before taking a finishing cut either when turning work held in a chuck or when boring or reaming.

FACE PLATES.

275. To hold work on face plate. — Some work can be clamped to a large face plate and machined more accurately and conveniently than if held in a chuck. The work *B* is clamped to face plate *A*, Fig. 176, by clamps *C* and *C′* and bolts *D* and *D′*.

If a finished surface is to be clamped against a face plate or other finished surface, insert a sheet of paper between to prevent slipping.

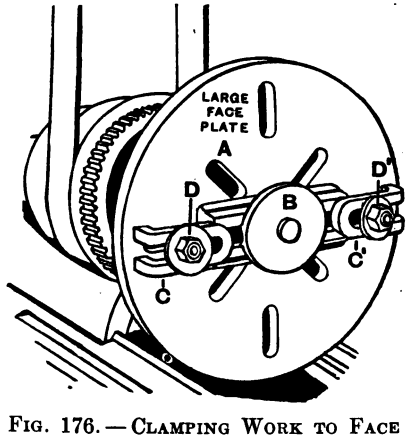


FIG. 176. — CLAMPING WORK TO FACE PLATE.

Digitized by Google

276. To clamp an engine crank to face plate, Fig. 177. — To face plate *A* crank *B* is bolted by bolts *C* and *C′* and clamp

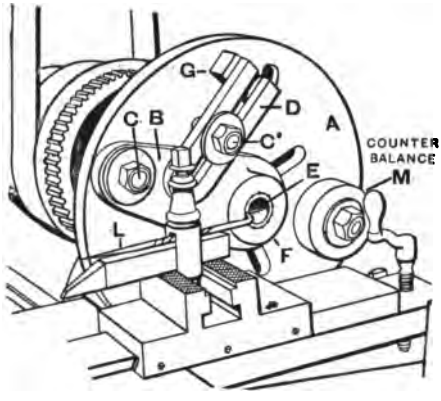


FIG. 177. — TESTING LOCATION OF ENGINE CRANK ON FACE PLATE.

D, in order to bore out hole *E* and turn and face hub *F*. Before clamping to face plate, plane the crank on its face and line out the holes, as at *H* and *I*, Fig. 178. Describe

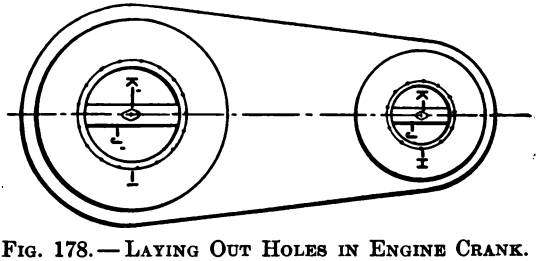


FIG. 178. — LAYING OUT HOLES IN ENGINE CRANK.

circles of the required diameter around the cored holes the proper distance apart for the crank throw. To provide centers for circles, drive pieces of wood into holes to form bridges, as at *J* and *J′*. Turn down the corners of a piece of tin, and drive it into the center of the bridge, as at *K* and *K′*. Rotate lathe by hand and move crank by rapping until circle is true to axis of rotation when tested with scriber *L*, Fig. 177, then clamp crank hard to face plate.

Digitized by Google

277. A counter weight to balance work is bolted to face plate at *M* in order to balance the eccentrically placed work and insure smooth running and accuracy.

278. To hold work with an angle plate. — Angle plate *A*, Fig. 179, is a useful fixture for various machine tools. It is planed all over with the faces at right angles (90°). It is bolted to face plate *B*, and pillow block *C* is clamped to

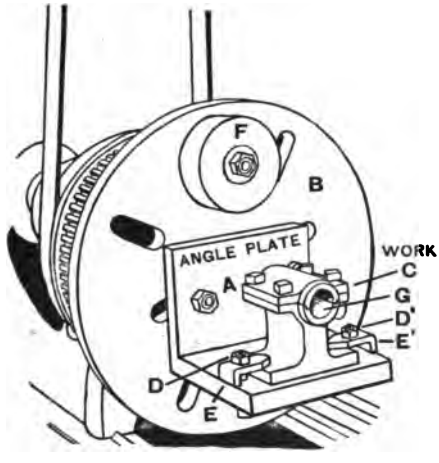


FIG. 179. — USE OF ANGLE PLATE CLAMPED TO FACE PLATE.

inside surface by bolts *D* and *D'* and clamps *E* and *E'*. *F* is a counterbalance. Before boring, the pillow block has had its base planed and the cap fitted and screwed on. A circle of proper diameter is described around the cored hole and center punched. The angle plate and work must be adjusted until this circle runs true, after which hole *G* may be bored and reamed.

279. The lathe axis indicator, shown at *A* in Fig. 180, is used to test the axial truth of a center punch mark on work held in a chuck or bolted to a face plate such as the engine crank shaft center fixture *B*, which is laid out and marked at *C* and *D* to be drilled, bored, and reamed exactly at these points.

Digitized by Google

The fixture is bolted to the face plate with mark *C* approximately central and counterbalance *E* opposite. The point of indicating needle *F*, which is pivoted in the universal joint *G*, is placed in center mark *C*.

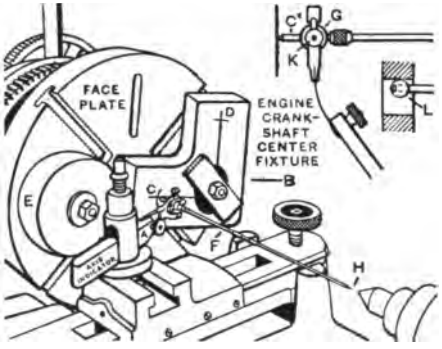


FIG. 180. — TESTING AXIAL TRUTH OF CENTER PUNCH MARK WITH INDICATOR.

As the lathe is rotated by hand, the needle point at *H* will revolve in a circle, exaggerating the error at *C*. The fixture is moved by tapping until needle point *H* remains stationary, when the lathe is rotated.

This indicator may be used to locate holes as well as center marks by first tightening nut *K*, which converts the universal into a single joint with a vertical movement, and then placing spherical head *L* upon point *C*, and using it against the upper wall of the hole. In this way the fixture may be used also to test the truth of a live center or a shaft turning on centers.

Attention. — Paper is sometimes placed between the face plate and smooth work to prevent the work slipping.

CHUCKING.

280. A method of drilling and reaming. — In chucking, the drill is stationary, while the work rotates. In drilling, the drill rotates and the work is stationary. Boring *is the enlarging* of a hole with a boring tool, or boring bar. Chucking is used where it would be impractical to drill.

Digitized by Google

281. Chucking with twist drill in engine lathe, Figs. 181 and 182.

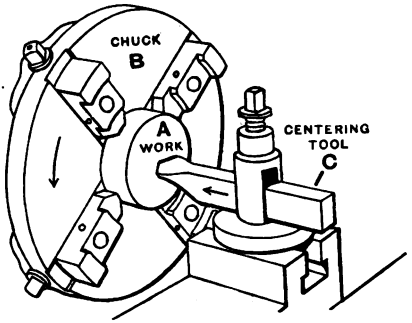


FIG. 181.—CUTTING A CONICAL CAVITY AXIALLY TRUE TO START TWIST DRILL.

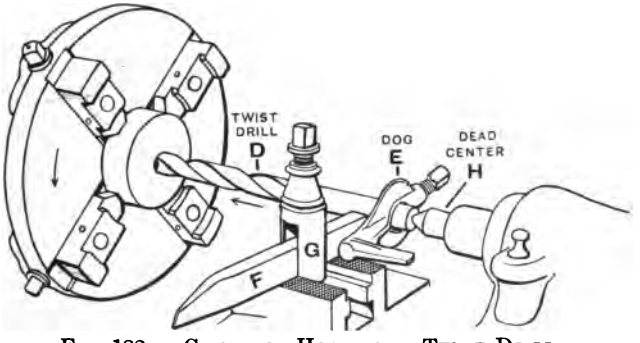


FIG. 182.—CHUCKING HOLE WITH TWIST DRILL.

SCHEDULE OF OPERATIONS AND TOOLS.

Set dead center in approximate alinement.

Drill, $\frac{1}{100}$″ to $\frac{1}{64}$″ small. Hand reamer, standard size.

1. Mount work *A* in independent chuck *B*, Fig. 181.
2. True up work. See § 266.
3. Cut cavity of same angle and diameter as drill, in center of work to start drill true, with graver, right side tool or, preferably, centering tool *C*, Fig. 181.
4. Fasten dog *E* to shank of twist drill *D*, Fig. 182.
5. Place tool *F* in tool-post *G*.
6. Hold *F* against dog by long. feed handle with left hand, to prevent drill feeding away from center *H*.
7. Feed footstock spindle with right hand.
8. Help carriage along with left hand.
9. When drill breaks through hold it hard against dead center. To ream hole, see § 294.

Digitized by Google

Attention. — Solid work, especially steel, is chucked with a two-lip twist drill. Three and four-groove drills are used for cored work (castings) or to follow a two-groove drill. Smooth holes may be made with a drill $\frac{1}{100}''$ to $\frac{1}{64}''$ small and a hand reamer; but better results are obtained by also using a fluted chucking reamer .005″ small before the hand reamer. See § 297.

282. Drill holder and steady rest, *A*, Fig. 183, is used with taper shank drills. Rest *B* and guide bushing *C* are used to center, steady, and guide three-groove twist drill *D* in work *E* which is a cored casting held in chuck *F*. Extra guide bushings are supplied to fit drills of different diameters.

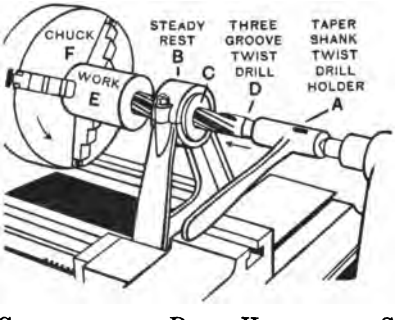


FIG. 183. — CHUCKING WITH DRILL HOLDER AND STEADY REST.

283. Flat chucking drills for chucking in an engine lathe, for either cored holes or solid work, Fig. 184. Large counter-

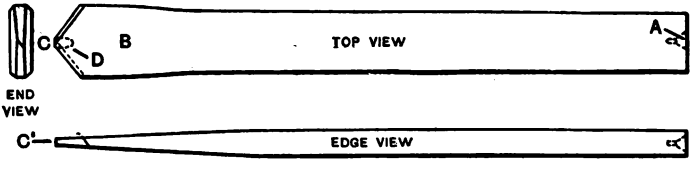


FIG. 184. — FLAT CHUCKING DRILL.

sink *A*, Fig. 184, provides a firm bearing on dead center. End *B* is $\frac{1}{64}''$ smaller in diameter than the chucking reamer.

Digitized by Google

Point CC' is central and either thinned or grooved on both sides, as at D. To give the cutting lips some rake, grooves may be ground above them, one of which is shown

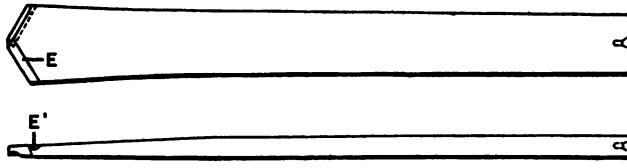


FIG. 185. — GROOVED-LIP FLAT CHUCKING DRILL.

at EE', Fig. 185. A better way is to twist the lips as at F and G, Fig. 186.

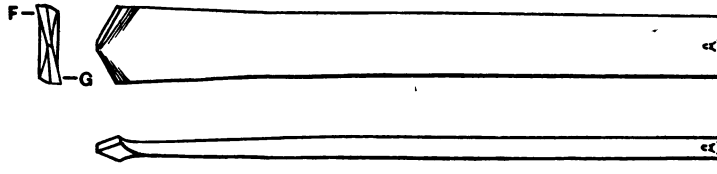


FIG. 186. — TWISTED-LIP FLAT CHUCKING DRILL.

284. Flat chucking reamers, Fig. 187. — The cutting edges are AB and CD. Head E is from .005″ to .010″ under size to allow for hand reaming.

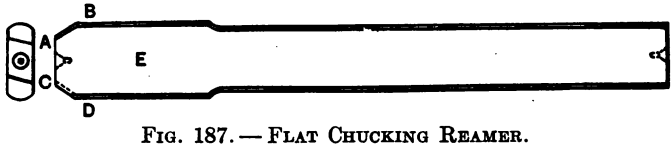


FIG. 187. — FLAT CHUCKING REAMER.

Attention. — Reamed holes have a very slight taper and the end the reamers enter is always the larger; therefore drill and ream work from the side into which the shaft is to be fitted.

285. Chucking with a flat drill and chucking reamer in an engine lathe. Fig. 188.

Digitized by Google

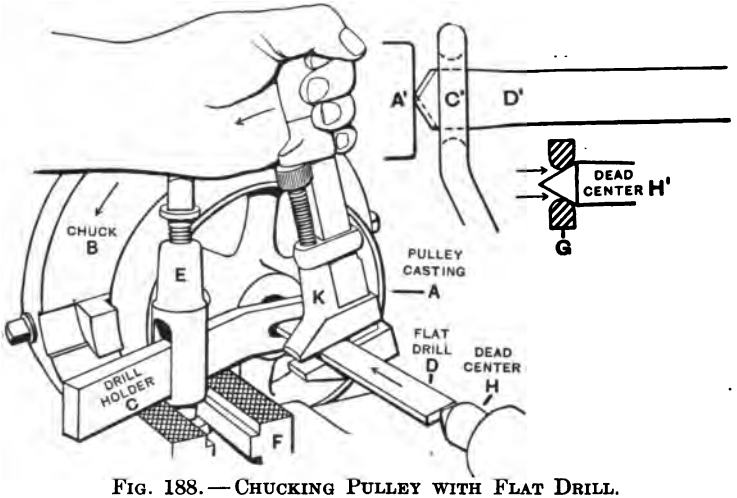


FIG. 188. — CHUCKING PULLEY WITH FLAT DRILL.

SCHEDULE OF OPERATIONS AND TOOLS.

Set dead center in approximate alinement.
Drill, $\frac{1}{64}''$ to $\frac{1}{32}''$ small. Chucking reamer, $\frac{1}{100}''$ small.
Drill rest. Monkey wrench. Hand reamer, standard size.

1. Mount pulley *A* in independent chuck *B*.
2. True up by inside of rim.
3. Clamp drill holder *C* in post *E* to tool block *F*.
4. Set holder with dead center *H'* exactly in middle of slot as at *G* and with drill approximately central as *C'*, *D'*.
5. Set holder near work as at *C'*, *A'*.
6. Place point of drill central against hub *A'*.
7. Place dead center *H* in other end.
8. Pull wrench *K* forward to pinch drill in slot.
9. Start lathe at speed for twist drills.
10. Feed spindle until drill cuts half the depth of its point.
11. Remove wrench. Feed drill rapidly.
12. Hold drill back on dead center with left hand, when point breaks through.
13. Stop lathe when through.
14. Place flat chucking reamer in slot as in operation 4.
15. Hold with wrench until reamer is started. When through stop lathe. To ream hole, see §§ 289, 293, 294.

Attention. — If slot in holder is not at height of dead center, the drill or reamer will cut large and may spoil work. If drill moves sidewise when starting, replace wrench and repeat. Drill must cut true before reaching full diameter. For large holes or cored holes, use two or three flat drills of increasing diameters.

Digitized by Google

REAMING.

286. Reamers are used for sizing, smoothing, and standardizing straight and taper holes. See Taper Reamers, § 297, and *Advanced Machine Work.*

There are two general classes: chucking or roughing reamers, used in a machine, and finishing reamers, used by hand or power. The usual amount for the finishing reamers to cut is .005″ to $\frac{1}{64}$″ for cast iron, and .005″ to .010″ for steel and brass.

287. Irregularly spaced teeth. — To prevent chattering, reamer teeth or blades are spaced progressively wider as in Fig. 189, *A* to 1 to the right, and from 1 to *A* to the left.

The clearance *G* is given the teeth or lands to relieve the cutting edge. The point of a hand reamer is slightly tapered a distance equal to its diameter, to enter the hole. The shank, *H*, Fig. 191, is ground .001″ small, to prevent binding in hole.

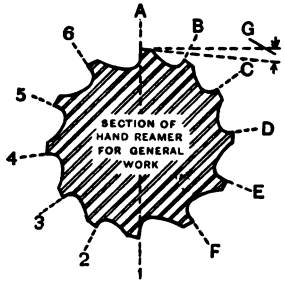


FIG. 189. — TEETH OF HAND REAMER IRREGULARLY SPACED.

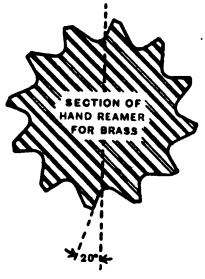


FIG. 190. — TEETH OF HAND REAMER WITH NEGATIVE RAKE.

288. To ream brass, the face of teeth is inclined 20° back of radial (negative rake) to prevent chattering, as in Fig. 190. See Broaching, *Advanced Machine Work.*

Attention. — Cast iron and brass are reamed dry; steel and wrought iron with oil.

Warning. — To ream thin work in a vise use a jig similarly constructed to the tapping jig, § 528. To ream thin work in

Digitized by Google

a vise *without* a jig and also without chattering and spoiling the work, the length of hole should be about twice the diameter of the reamer. See § 293.

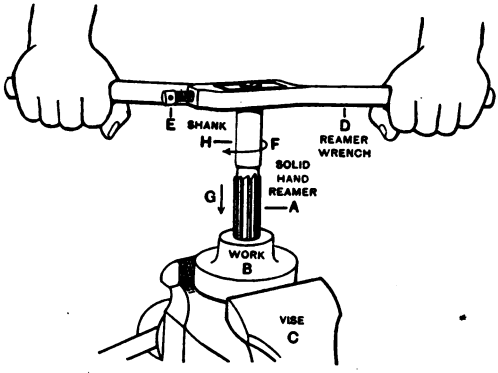


FIG. 191.—REAMING IN VISE.

289. Hand reaming work held in vise. Fig. 191.

SCHEDULE OF OPERATIONS AND TOOLS.

Hole in flange casting drilled or drilled and reamed .005″ to $\frac{1}{64}$″ small.

1. Chuck work *B* in engine lathe then fasten firmly in vise *C*. 2. Place adjustable reamer wrench *D* on square end of reamer *A* and fasten with thumb screw *E*.	3. Rotate wrench rapidly in direction of arrow *F* and at same time press downward in direction of arrow *G*, continue rotating and pressing downward until reamer passes clear through the casting.

Attention. — Hand reamers should never be rotated backward, as it quickly destroys their cutting edges, and they should be used vertically when the nature of the work will permit.

Note. — Before placing reamer in hole, see that there are no burrs on shank *H*, which would be likely to scratch the reamed surface as the reamer is passed through the hole.

Digitized by Google

290. To ream a hole by hand in a vertical drilling machine, Fig. 192.

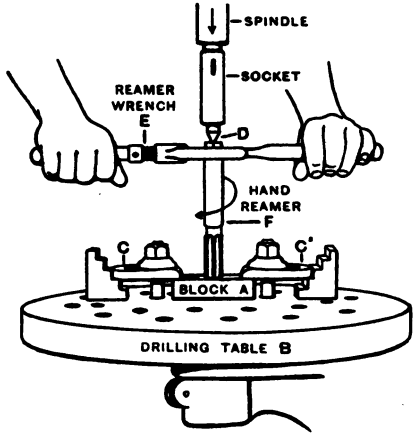


FIG. 192. — HAND REAMING IN A VERTICAL DRILLING MACHINE.

SCHEDULE OF OPERATIONS AND TOOLS.

Hole in work drilled .005″ to $\frac{1}{64}$″ small.

1. Place drill in socket and the socket in spindle.
2. Unclamp table at swivel and column.
3. Adjust table until drill will pass down through any convenient hole and reclamp.
4. Clamp block *A* to table *B* with clamps, bolts, and step blocks *C*, *C'*.
5. Drill hole with reamer drill. See Accurate Drilling, §§ 462–470.
6. Insert center *D* and place wrench *E* and reamer *F* in place.
7. Rotate reamer in direction of arrow, following with center *D*.

291. Adjustable reamers. — *A*, Fig. 193, may be adjusted to compensate for wear and to ream special-size holes.

292. Reaming stand *B*, Fig. 193, is more convenient than a vise. It consists of an independent chuck *C*, supported by column *D*. The work *E* is firmly gripped in the jaws *F* by operating the handle *G*.

Digitized by Google

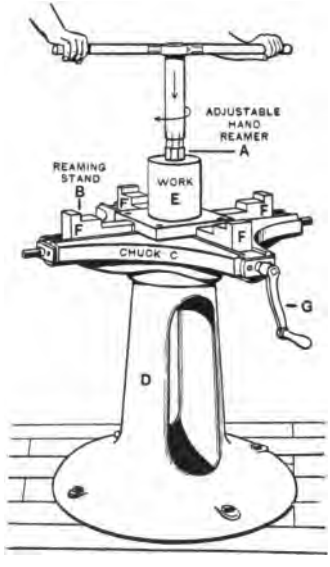


FIG. 193. — HAND REAMING IN REAMING STAND.

293. To ream work by hand in the lathe with a hand reamer, Fig. 194.

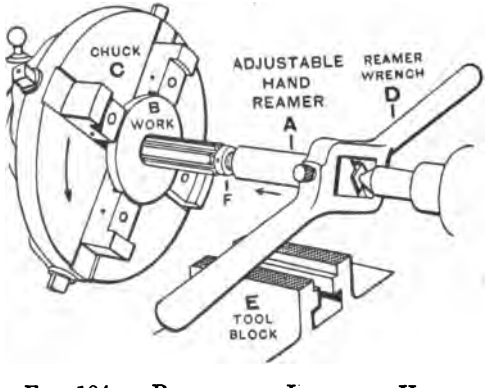


FIG. 194. — REAMING IN LATHE BY HAND.

Digitized by Google

SCHEDULE OF OPERATIONS AND TOOLS.

Hole in blank drilled or bored .005″ to $\frac{1}{64}$″ small.
Set dead center in approximate alinement.

1. Place reamer *A* in drilled or bored hole in work *B* held in chuck *C*.	2. Place arm of wrench *D* against tool block *E*. 3. Pull belt downward and follow with dead center.

Attention. — Carefully follow reamer with dead center to prevent spoiling reamer and work.

On thick work, start hand reamer in above manner in lathe, and then take both work and reamer to reaming stand or vise for finishing. See § 289.

294. Fluted chucking reamer, Fig. 195, is obtainable in standard sizes or .005″ small, to be followed by standard hand reamer. This class of reamers has its points always beveled, and some points are slightly tapered the same as on hand reamers. Chucking reamers are also obtainable with taper shanks to fit drill sockets.

295. To ream in lathe by power with fluted chucking reamer, Fig. 195. — Fasten dog *D* to shank of reamer *A* and insert reamers in drilled hole of work *B*, held in chuck *C*, allowing dog *D* to rest on tool block *E*.

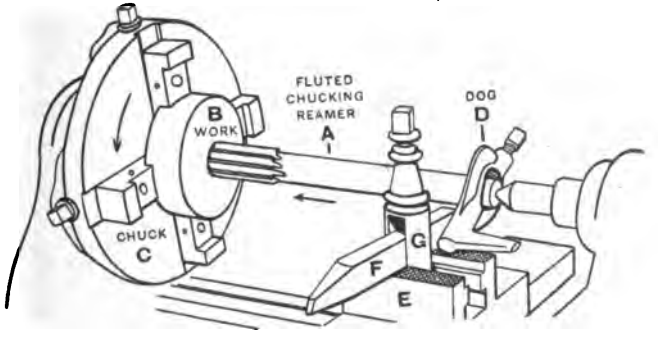


FIG. 195. — REAMING GEAR BLANK IN LATHE BY POWER.

Digitized by Google

SCHEDULE OF OPERATIONS AND TOOLS.

Hole in casting drilled $\frac{1}{16}''$ to $\frac{1}{64}''$ small.
Set dead center in approximate alinement.

1. Place reamer *A* in drilled hole in work *B* held in chuck *C*. Fasten dog *D* to shank of reamer and rest on tool block *E*. 2. Fasten tool *F* in post *G* and adjust to prevent reamer drawing off dead center.	3. Use power and help carriage along by hand. Use about two-thirds the speed recommended for twist drills of same diameter. To hand ream, see § 293.

296. Rose chucking reamer, Fig. 196, is made with either straight or helical flutes, with cutting teeth on end only. It will not produce as smooth a hole as a hand or fluted chucking reamer and is not used when a smooth hole is required. See § 295. It is obtainable in standard sizes and also with taper shank to fit drill sockets.

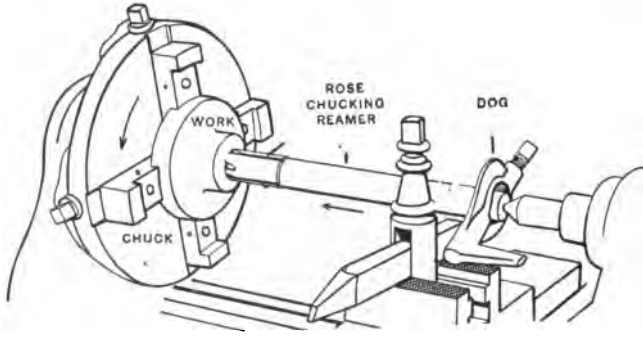


FIG. 196. — REAMING FLANGE CASTING BY POWER IN LATHE.

297. Broach reamers for small taper pins. — Collars, *A*, and similar work are often fastened to shafts by drilling, reaming, *B*, and pinning, *C*, Fig. 197.

Broach reamers may be supplied with a handle, as at *E*, and used by hand to enlarge holes and to remove burrs. They are obtainable in tapers of $\frac{1}{8}''$ and $\frac{3}{16}''$ per foot, and in sizes from No. 1 to No. 70 (drill and wire gage sizes). Taper pin

Digitized by Google

fluted reamers and taper pins having a taper of $\frac{1}{4}''$ per foot are used for pinning or doweling, and are obtainable in sizes from $\frac{1}{8}''$ to $1\frac{1}{4}''$. Each reamer overlaps the next smaller size.

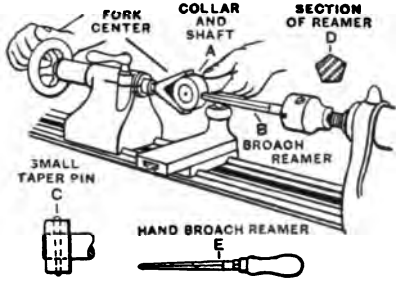


FIG. 197. — REAMIMG TAPER PIN HOLES WITH BROACH REAMER IN SPEED LATHE.

Attention. — To avoid breaking taper reamers, feed slowly, oil freely, and withdraw frequently to remove chips.

Digitized by Google

CHAPTER XI.

MANDRELS OR ARBORS. MANDREL MAKING. TURNING AND FINISHING FLANGES. TURNING AND FINISHING PULLEYS. POLISHING LATHE WORK.

MANDRELS OR ARBORS.

298. A mandrel, often called an arbor, is pressed, driven, or threaded into work to provide centers so that it may be machined.

An arbor is a shaft used to carry a cutting tool, as a milling machine arbor, a saw arbor, etc.

Three classes of mandrels are used: solid, expanding, and built-up. See Nut Mandrel, § 256.

299. Standard solid mandrel, *AB*, Fig. 198, is made of

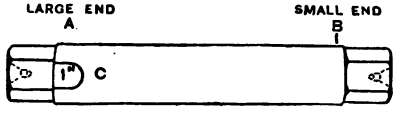


FIG. 198. — MANDREL OR ARBOR.

tool steel, hardened and ground. See Table of Mandrels, § 652. The size is stamped at large end, as at *C*. A portion

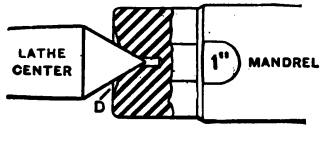


FIG. 199. — SECTION OF MANDREL END.

of each end is reduced and flattened to receive the driving dog. The ends are recessed as at *D*, Fig. 199, around the countersinks to protect them from injury.

Digitized by Google

300. Expanding mandrel. — Fig. 200 consists of shaft *A* having uniformly tapered slots as at *B*. Sleeve *C* guides tapered jaws *D* (see *D'*) while they are being adjusted to the work by the sliding of the shaft *A*.

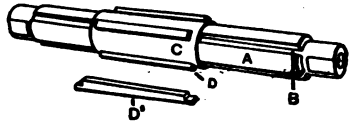


FIG. 200. — EXPANDING MANDREL.

301. Bridges in hollow castings *A*, Fig. 201, are often cast across the ends as at *B*, to provide for center holes.

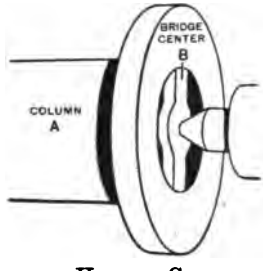


FIG. 201. — BRIDGE IN HOLLOW CASTING FOR CENTER HOLE.

302. Revolving dead center for pipe turning. — To square or turn small pipe, mount upon ordinary lathe centers. For large pipe or cored work, a special large dead center is needed,

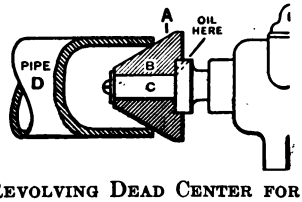


FIG. 202. — REVOLVING DEAD CENTER FOR PIPE TURNING.

preferably one that will revolve as at *A*, Fig. 202, in which cone *B* revolves upon shank *C*. A solid live center or chuck may be used at other end.

303. Built-up and special mandrels. — To face the ends of small engine cylinders or similar work, a built-up mandrel is used. See *Advanced Machine Work*. To face the ends of a piece of pipe

Digitized by Google

or cored work, make a mandrel by drilling and tapping three or four holes around each end of a stiff shaft and inserting adjusting screws to bear against the inside of work.

For large shells, tubes, etc., castings called "spiders" are used consisting of arms projecting from a hub, each arm supplied with a screw to bear against the work, and the hub supplied with set screws to fasten spider to mandrel.

A special mandrel may be made of any suitable piece of stock by centering, turning, and filing to fit the hole.

304. Soft hammers for driving mandrels into work are made of lead or rawhide. A steel hammer should never be used without protecting work with a block of wood, copper, or lead.

Attention. — Molds are obtainable for molding lead hammers.

305. Mandrel or arbor block.— Fig. 203 shows the relative position of block *A*, mandrel *B*, work *C*, and soft hammer *D* when driving a mandrel into the work.

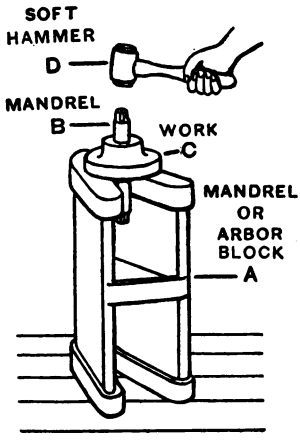


FIG. 203. — DRIVING MANDREL INTO FLANGE.

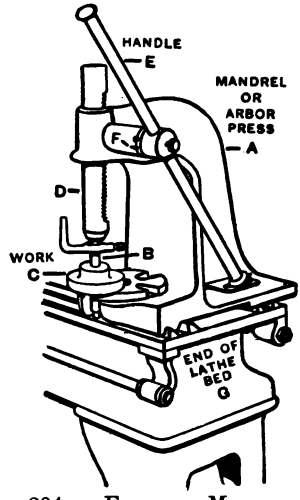


FIG. 204. — FORCING MANDREL INTO FLANGE WITH MANDREL PRESS.

306. Mandrel or arbor press, *A*, Fig. 204, is used to press mandrel *B* into work *C* by forcing spindle *D* against the mandrel with handle *E*. Pointer *F* may be set to adjust mandrel for dupli-

Digitized by Google

cate pieces. The press may be bolted to a lathe as at *G*, or mounted on a stand. It may be used also for forcing fits.

307. Center mandrel for bottom holes. — Mandrels made to fit the hole in the live spindle of a lathe are used to hold caps, oil cups, bottom or blind nuts and similar work that must be machined. The projecting end is turned either to a slight taper to receive work with a plain hole, or threaded to hold threaded pieces.

308. Studs for driving large work instead of dog. — Work of large diameter, as a large pulley, may be driven by its arms with one or more studs fastened to the face plate of lathe.

Attention. — Oil mandrel before pressing or driving it into work. Take light cuts when using small mandrels to avoid springing of mandrel and chattering, producing irregular tool marks.

MANDREL MAKING.

309. To make a standard lathe mandrel, Fig. 205, involves practice in squaring, turning, milling, hardening, tempering, and cylindrical grinding.

If not equipped with cylindrical grinding machine, harden and temper ends only, turn body slightly taper and file to fit standard hole.

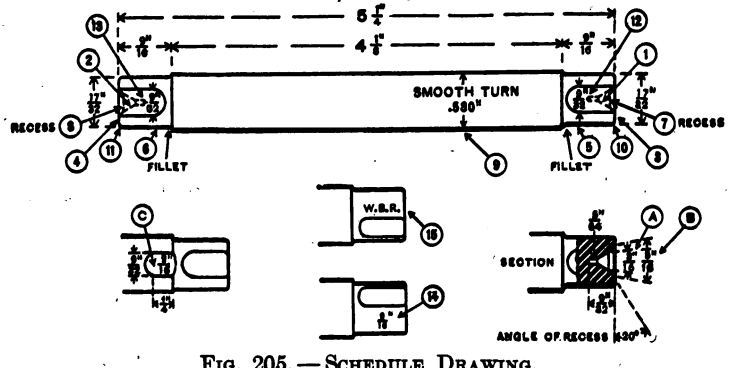


FIG. 205. — SCHEDULE DRAWING.

310. To prepare a $\frac{9}{16}$" standard mandrel blank for hardening, tempering, and grinding, Fig. 205.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.

Stock, annealed carbon steel $\frac{1}{16}''$ large.

True live center. Set dead center in approximate alinement.

Machine dry or use lard oil.

Use high-speed steel cutting tools. See *Exception*, p. 59.

Operations.	Machines, Speeds, Feeds.	Tools.
Center to $\frac{3}{16}''$, (1), (2). See (A).	Centering machine. Drill 1000 R. P. M., countersink 400 R.P.M.	$\frac{5}{64}''$ drill, or combination drill and countersink, 60°. Lard oil.
Rough square to $5\frac{1}{4}'' + \frac{1}{64}''$, (3), (4).	Engine lathe 12″ to 16″, 2d or 3d speed, or 35 F.P.M. Hand feed.	Dog, side tool, 30° rake, or holder and cutter, calipers, rule.
Recenter to $\frac{3}{16}''$, (1), (2).	Speed lathe, 2d or 3d speed.	Drill, countersink.
Finish square to length, (3), (4).	Engine lathe, 3d or 4th speed, or 50 F. P. M.	
Rough turn reduced portions to $1\frac{7}{32}'' + \frac{1}{64}''$ (5), (6), one or two cuts. Do not square shoulders but leave fillets as shown to avoid cracking in hardening.	2d or 3d speed, or 30 F. P. M. Medium power feed — 80 to 1″.	Copper under set screw of dog, diamond-point tool, or holder and cutter, 30° rake, calipers, rule.
Recess countersinks to dimensions $\frac{5}{16}''$. Angle 20°, (7), (8). See (B).	3d or 4th speed, or 50 F. P. M.	Side tool or special countersink. See Fig. 23. Lard oil.
Recenter to $\frac{3}{16}''$, (1), (2).	Speed lathe, 2d or 3d speed.	Drill, countersink.
Finish turn reduced portions, (5), (6), one cut.	Engine lathe, 3d or 4th speed, or 50 F.P.M. Fine power feed — 140 to 1″.	
Smooth turn body to .580″. Turn half of length and reverse (9), one or two cuts.	2d or 3d speed, or 30 F. P. M. Medium power feed — 80 to 1″.	Micrometer.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.
Concluded.

Operations.	Machines, Speeds, Feeds.	Tools.
Round corners, (10), (11).	Speed lathe, 3d or 4th speed, or 200 F.P.M.	Graver.
File reduced portions.	3d or 4th speed, or 175 F.P.M.	8″ or 10″ mill bastard file.
Polish reduced portion, also ends and recesses except countersinks, (5), (6). See §§ 319–322.	highest speed.	Lard oil, 90 and 120 emery cloth, polishing stick.
Mill or file and polish flats, (12), (13).	Milling machine, 3d speed, medium power feed.	*To mill*, use milling machine vise, parallel piece, 1″ end mill, rule, lead hammer, oil. *To file*, use 8″ or 10″ hand smooth and 5″ half-round, 2d cut files. *To polish*, use oil, 90 and 120 emery cloth.
Stamp size $\frac{9}{16}$″, (14), or mill or file flat and stamp size at (C).	Vise.	$\frac{1}{16}$″ steel figures, $\frac{1}{8}$″ chisel, hammer, copper jaws.
Stamp name or initials (15).	.....	Steel name stamp.
Harden (test with file), leave ends black, or polish and temper ends to dark straw. See *Elements of Machine Work*.	Gas furnace or forge. speed lathe. Oil, tempering furnace.	Tongs, water, brine, or sperm oil, oily waste, dead smooth file, red-hot iron rings or oil, temper.
To lap center holes, see *Attention* below.		
To grind, see § 422.		

Attention. — The hand tools may be carbon steel.

See Table of Mandrels, § 657.

Note. — For very accurate work, center holes in mandrels are lapped smooth and true before grinding, but this process is often omitted on mandrels for general work. To make a lap, hold a piece of

Digitized by Google

¼″ copper rod in chuck of speed lathe and turn with graver or planisher to 60°. Test with center gage. Charge lap with 120-grain emery and oil. Mount mandrel on dead center and on lap. Run lathe at highest speed, revolve mandrel slowly backward, and relieve dead center occasionally.

For further information on Lapping, see *Advanced Machine Work*.

TURNING AND FINISHING FLANGES.

311. To clamp carriage to face large work in engine lathe. — Feed tool to its cut by hand, then clamp carriage with binding screw and use power cross feed.

In the absence of power cross feed and clamp, throw feed belt off or feed gears out of mesh. Run tool close to work, tighten long. feed friction knob, revolve feed shaft by hand until tool takes required cut and then feed tool inward or outward with hand cross feed.

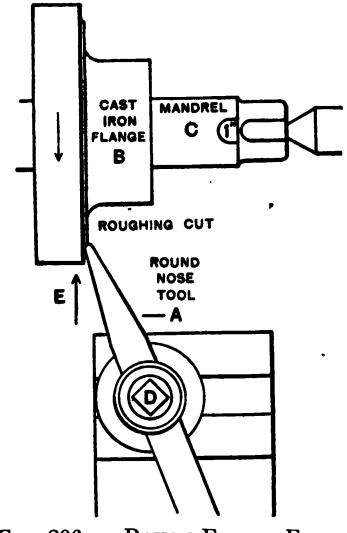


FIG. 206. — ROUGH FACING FLANGE.

312. To rough turn face of cast-iron flange, Fig. 206. — Use round-nose tool *A* for roughing flange *B* on mandrel *C*. Fasten tool in post *D*, and clamp carriage. Feed in direction of arrow *E* to outer edge of fillet.

Digitized by Google

313. To finish and scrape face of cast-iron flange, Fig. 207, **to prepare for polishing.** — Take light finishing cuts with

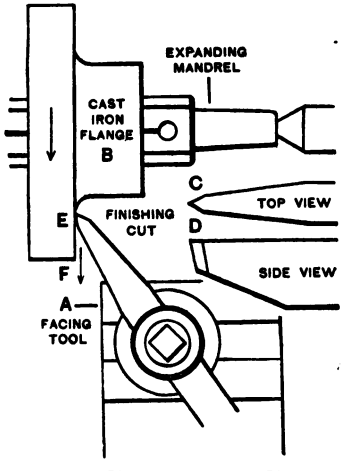


FIG. 207. — FINISH FACING FLANGE.

facing tool *A* ground to 60°, as at *C* and *D*, at medium high speed. Grind and oilstone top face of tool. Set it to drag a little as at *E* and feed in direction of arrow *F*. Use scraper

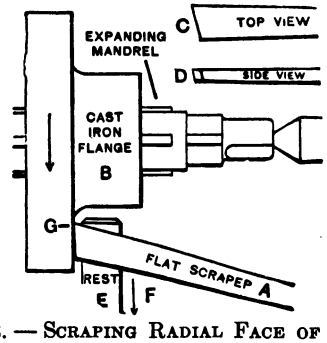


FIG. 208. — SCRAPING RADIAL FACE OF FLANGE.

A, Fig. 208, with cutting edge *C D* ground straight, or slightly convex, at medium speed. Hold scraper on rest *E* and move in direction of *F* with point dragging a little as at *G*. A piece of leather placed under scraper will often prevent chattering.

Digitized by Google

314. To scrape inside rounds or fillets to the desired curvature. First shape with a round-nose lathe tool. Then use a round-nose hand tool, *A*, Fig. 209, to prepare filleted corner of flange *B* for polishing. *C* and *D* show point and clearance of tool. To avoid chattering, curvature of end *E* should be greater than that of fillet for clearance and the tool should be held firmly on rest. A round or half-round file is sometimes used to smooth a fillet.

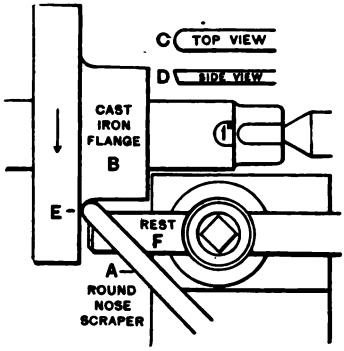


FIG. 209. — SCRAPING FILLET IN FLANGE.

Attention. — Cylindrical surfaces are more easily prepared for polishing than radial and curved surfaces. On small work a light finishing cut is taken with a fine feed, and on very large work a light finishing cut is

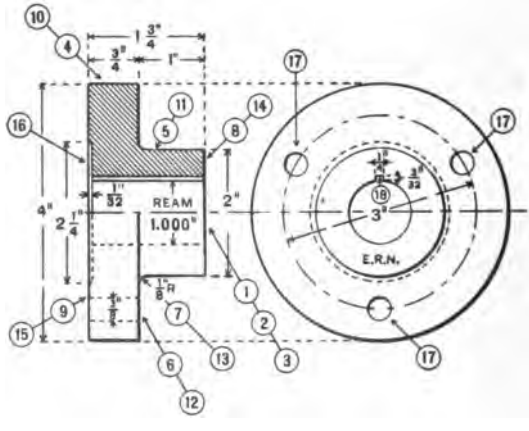


FIG. 210. — SCHEDULE DRAWING OF MAKING A CAST-IRON FLANGE.

taken with a broad-nose tool and a coarse feed, and the surfaces filed to erase the tool marks.

315. To make a cast-iron flange finished all over, Fig. 210.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.

Stock, iron casting $\frac{1}{8}''$ large.

True live center. Set dead center in approximate alinement.

Use high-speed steel cutting tools. See *Exception*, p. 59.

Operations.	Machines, Feeds, Speeds	Tools.
Snag casting. Mount in chuck and adjust until hub runs true. Drill and rough ream hole, (1), (2).	Engine lathe, 12″ to 16″. 2d or 3d speed, or 115 R.P.M. Hand feed.	File, independent chuck, chalk, drill rest, flat drill, $\frac{11}{32}''$, flat reamer, $\frac{99}{100}''$.
Finish ream hole by starting reamer in lathe, pull belt by hand, complete at vise, (3). See § 289.	Reaming stand or vise. See §§ 305, 306.	1″ standard hand reamer, reamer wrench.
Oil mandrel and press into hole in direction hole was reamed. See §§ 305, 306. Mount on centers and rough turn $\frac{1}{64}''$ oversize, (4), (5).	Mandrel press. Engine lathe, 4th speed. Back gears in, or 35 F. P. M. Medium power feed — 80 to 1″.	1″ mandrel, holder and cutter or diamond-point tool, 15° rake, calipers, rule.
Rough face, (6); feed inward; rough fillet, (7); rough face, (8); reverse work on centers, then rough (9).	4th speed. Back gears in, or 35 F.P.M. Hand or medium power feed.	Round-nose tool or holder and cutter, 15° rake.
Finish turn, (10), (11).	3d or 4th speed. Back gears in, or 50 F.P.M. Fine power feed — 140 to 1″.	Holder and cutter or diamond-point tool, 15° rake, calipers, rule.
File, (10), (11).	3d speed, or 175 F.P.M.	8″ or 10″ mill bastard file, file card.
Finish face with cuts .002″ to .003″ deep and scrape, (12).	3d or 4th speed, or 225 F.P.M.	Facing tool, flat scraper.
Finish face and scrape, (13).	3d or 4th speed, or 225 F.P.M.	Round-nose tool, round-nose scraper.
Finish face and scrape, (14).	3d or 4th speed, or 225 F.P.M.	Facing tool, flat scraper, 2 rules to measure length of hub.
Finish (do not scrape or polish), (15).	3d speed, or 60 F.P.M.	Facing tool, calipers, rule.
Cut recess $\frac{1}{32}''$ deep. Set side tool at 45°, feed outward, (16).	3d speed, or 50 F.P.M. Hand or medium power feed — 80 to 1″.	Side tool, calipers, rule.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.
Concluded.

Operations.	Machines, Feeds, Speeds.	Tools.
Polish, (10), (11), (12), (13), (14). See §§ 319 – 321.	Speed lathe, highest speed.	Polishing stick, 60, 90, 120, emery cloth, oil.
Jig drill, and ream holes, (17).	2-spindle drilling machine. Drill 600 R. P. M. Power reamer, 400 R.P.M. Hand feed.	Box jig, $\frac{23}{64}$″ drill, $\frac{3}{8}$ power reamer.
Cut keyway central, (18).	Planer and vise or key seating machine.	Keyway tool holder and cutter, $\frac{1}{4}$″ wide.
Stamp name or initials in recess, as E.R.N.		Steel name stamp, hammer.

Attention. — Finishing cuts 10 and 15 may be omitted until flange is keyed to its shaft. The hole may be chucked with a $\frac{23}{64}$″ twist drill or, if cored, with a three or four-groove drill, and hand reamed (1), (3). The hand tools may be carbon steel.

Note. — Bushings, collars and step pulleys or any other work mounted on a mandrel, with one, two, or more diameters, are machined in the same general manner as the flange in Fig. 210.

TURNING AND FINISHING PULLEYS.

316. The taper or crowning on the face of pulleys, ranges from $\frac{1}{4}$″ per foot for large pulleys to $\frac{3}{4}$″ per foot for small pulleys. Tapered-face pulleys are made commercially in a pulley lathe with two cutting tools operating at same time, — one on the front, the other on the back of machine, or in turret lathes with special tools.

The face of the pulley is often crowned in a forming lathe or ground in special grinding machines from the rough casting. The hubs of pulleys used in combination, as tight and loose pulley mechanism, are squared.

Pulley *A*, Fig. 211, is turned in an engine lathe. The rim is faced as at *B*. The face is tapered as at *C*. The rim may

Digitized by Google

be chamfered as at *D*, or rounded with a right-and-left forming tool as at *E*, *E'*.

The set-over of footstock for ordinary tapers is forward, as at *A*, Fig. 212, and for pulleys it is backward as at *B*.

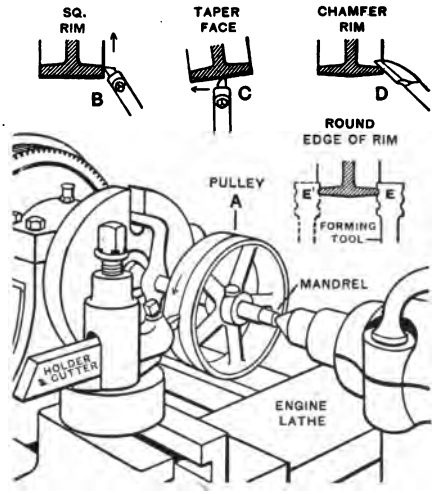


FIG. 211. — PULLEY TURNING IN ENGINE LATHE.

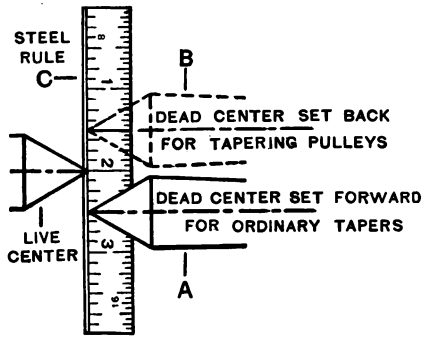


FIG. 212.

Digitized by Google

317. To make a pulley 5″ in diameter.

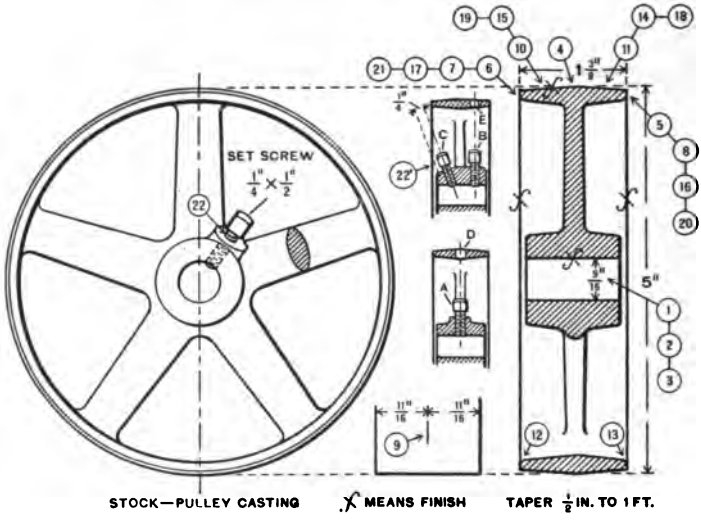


FIG. 213. — SCHEDULE DRAWING.

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.

Stock, pulley casting. Surfaces to be finished $\frac{1}{16}$″ large.
True live center. Set dead center in approximate alinement.
Use high-speed steel cutting tools. See *Exception*, p. 59.

OPERATIONS.	MACHINES, SPEEDS, FEEDS.	TOOLS.
Snag casting. Mount in chuck and adjust until edge and inside of rim run true. Drill and rough ream hole, (1), (2).	Engine lathe, 12″ to 16″. 3d speed, or 400 R.P.M. Hand feed.	File, chuck, chalk, lathe centering tool, $\frac{53}{64}$″ twist drill, dog, chucking reamer, .005″, small.
Finish ream hole by starting reamer in lathe, pulling belt by hand, completing at vise, § 289, (3).	Reaming stand or vise. See §§ 305, 306.	$\frac{9}{16}$″ standard hand reamer, reamer wrench.
Oil mandrel and press into hole. See §§ 305, 306. Mount on centers and rough turn to 5″, (4), one or two cuts.	Mandrel press. Engine lathe, 3d or 4th speed, back gears in, or 35 F. P. M. Medium power feed — 80 to 1″.	Dog, standard mandrel holder and cutter or diamond-point tool, 15° rake, calipers, rule.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.
Concluded.

Operations.	Machines, Speeds, Feeds.	Tools.
Rough square edges of rim to $1\frac{3}{8}'' + \frac{1}{64}''$, (5), (6). Take equal cuts from each side.	3d or 4th speed, back gears in, or 50 F. P. M. Hand or medium power feed — 80 to 1″.	Holder and cutter or round-nose tool, calipers, rule.
Regrind tool, finish square edges to width, (7), (8). Take equal cuts each side.		
Chalk spot on face and make line midway between edges, (9).	.....	Chalk, divider - calipers or dividers, rule.
Set footstock back to turn taper of $\frac{1}{2}''$ per foot ($\frac{7}{64}''$ for $5\frac{1}{4}$ between centers) and set tool to touch line (9), and feed toward headstock. Reverse pulley, reset tool and repeat, (10), (11).	3d or 4th speed, back gears in, or 50 F.P.M. Fine power feed — 140 to 1″.	Holder and cutter or diamond-point tool.
Chamfer at 45° inside corners of rim, or round with forming tool, (12), (13).	2d speed, back gears in, or 25 F.P.M., hand feed.	Side tool, 15° rake.
File face, (14), (15). Scrape edges, (16), (17).	2d speed, or 175 F.P.M.	8″ or 10″ mill bastard file, graver.
Polish face and edges, (18), (19), (20), (21), See §§ 319, 321.	Speed lathe, highest speed.	60, 90, 120 emery cloth, polishing stick, oil.
Drill and tap hole for set screw, insert $\frac{1}{4}'' \times \frac{1}{2}''$ round-point set screw, (22).	Vertical drilling machine, 3d speed, hand feed. Tilting vise or fixture.	$\frac{3}{16}''$ extension drill, $\frac{1}{4}'' \times 20$ U. S. S., pulley tap, wrench, lard oil.
Stamp name or initial on end of hub.	.....	Steel name stamp, hammer.
Paint unfinished parts.	.....	Special black paint.

Attention. — Power chucking reamer may be omitted, as a good hole may be obtained with a drill and hand reamer.

When possible, place arms of pulley opposite chuck jaws. The hand tools may be carbon steel.

Digitized by Google

318. To locate set screws, Fig. 213. — If the boss is central as at *A*, drill body hole *D* in rim through which to tap hole *A*, and use a cup or round-pointed set screw. Where the boss is flat as at *B* and placed at one side, drill a body hole through rim *E*. If a body hole through rim is objectionable, hold pulley in a tilting vise or fixture and drill and tap hole at angle, as at *C*.

In large pulleys use two set screws or key.

POLISHING LATHE WORK.

319. Use a speed lathe for polishing but never an engine lathe when it can be avoided, as the gritty spatterings injure its bearings. Make exception only to work machined in a chuck of an engine lathe which must be polished before it is removed.

An effective polishing speed for emery, alundum, or carborundum cloth is between 5000 and 6000 F.P.M.; that is, a surface speed of about a mile a minute (5280 feet).

It is not always possible to obtain this high speed, as speed lathes are usually belted for a great variety of work, as drilling, hand turning, etc., and their highest speed will not give the maximum polishing speed for small work.

On work that is unbalanced, it is not desirable, and often dangerous, to polish at the maximum speed, as it will shake the lathe and the work may fly off the centers. It is best to use as high a speed as the nature of the work will permit, since the polishing can then be done with less labor and with less tendency to destroy the truth of the work.

320. Order of applying different numbers of emery cloth. — If the work is carefully finished, limited application of 60 and 90 will produce an effective polish; and if the work is finished extra fine, 90 will be sufficient. If a more brilliant polish is desired, use 120 and flour. For large work not given a fine finish it may be necessary to begin with 46 or 54.

Apply 60 emery first with hard pressure until all tool marks and scratches are removed and the pores in the metal have nearly disappeared. Then apply 90 emery with lighter pressure until all evidence of 60 is removed. If on applying

Digitized by Google

90, tool marks, deep scratches, or large pores appear, return at once to 60 emery. Follow this method in applying successive grades of emery.

Use lard oil on the emery cloth or work, distributing it with the fingers.

321. To polish flange, Fig. 214. — Oil speed lathe and shift belt to highest speed. Press mandrel in flange *A*, which has been carefully finished by turning, filing, and scraping. Fasten dog on mandrel. Mount and adjust on centers. Wrap a strip of No. 60 emery cloth *B* around end of wedge-shaped soft-pine stick *C*. Drop a little oil on emery cloth or flange and distribute with fingers. Start lathe and pivot stick on rest *D* which should be clamped from 2″ to 2½″ from the work.

Keep emery moving back and forth slowly along the work so that the marks will cross and recross each other, to avoid cutting grooves. Grades 60 and 90 will produce a good polish on a flange.

To polish radial face of flange, clamp rest *D* parallel to face and from 2″ to 2½″ distant. Apply emery cloth in same order

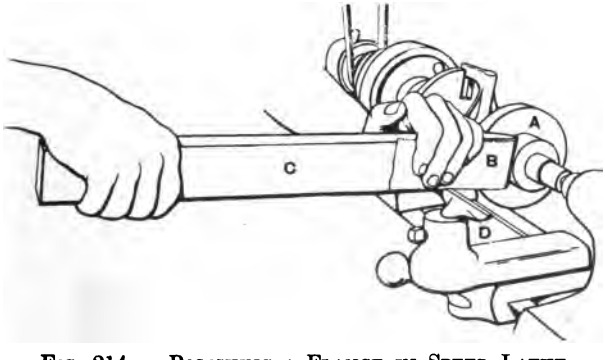


FIG. 214. — POLISHING A FLANGE IN SPEED LATHE.

as on hub. Keep emery moving slowly, advance and recede in short strokes toward the center to avoid cutting grooves, then recede in the same order. To polish the fillet, move emery cloth back and forth in short strokes following the curve.

Digitized by Google

322. To polish a shaft with wood polishing clamp, AA′ (Fig. 215), hinged by leather at end.— Oil speed lathe and shift belt to highest speed. Place dog on shaft with copper under set screw, and mount shaft on centers. Drop a little oil on emery cloth or shaft and distribute with the fingers. Wrap

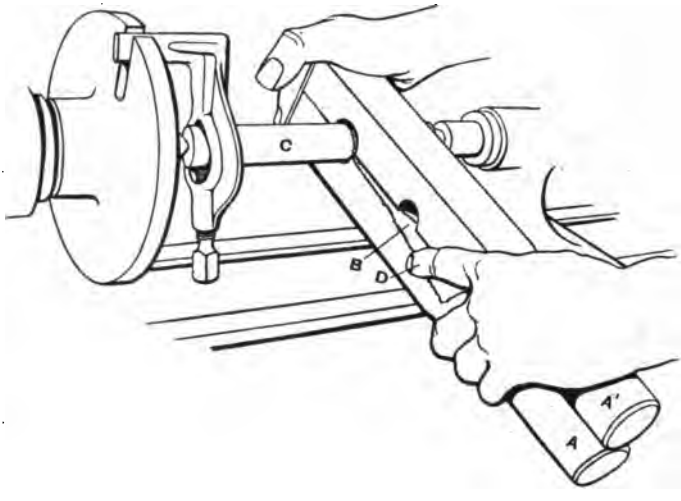


FIG. 215. — POLISHING A SHAFT IN SPEED LATHE.

emery cloth *B* around shaft *C* in one fold only, and hold end as at *D*, to prevent it from winding around shaft. Apply pressure with the left hand, and with the right move clamp back and forth in short strokes along shaft so that the marks will cross and recross each other. Move clamp continuously when in contact with revolving work, otherwise emery will cut grooves. Grades 60 and 90 will produce a good polish on a shaft.

323. To polish brass and copper, use a finer emery cloth than for steel or iron, as the material is softer. Start with a No. 90, continue with No. 120 flour, and crocus cloth. If a more brilliant luster is desired, the work may be buffed.

For Polishing and Buffing with Wheels and Belts, and Lacquering, see *Advanced Machine Work.*

Digitized by Google

Attention. — Use oil sparingly to avoid excessive spattering. Do not allow the emery cloth to slip off the edge of work as it will round the corners. Adjust mandrel, or work freely on centers, and occasionally loosen and oil dead center, as the heat generated will expand the work or mandrel and burn off the dead center.

To lay lines uniformly after work is polished, moisten a piece of worn emery cloth of fine grade with oil (or oil the work, distributing it with the fingers) and move the emery cloth slowly along the work.

Digitized by Google

CHAPTER XII.

HAND TURNING. MAKING MACHINE HANDLES. DRILLING, TAPPING, AND HAND THREADING IN SPEED LATHE. SPRING WINDING.

HAND TURNING.

324. Hand turning tools such as the graver, round-nose tool, etc., are forged from $\frac{1}{4}''$ to $\frac{3}{8}''$ square, hexagonal, and round, carbon steel, hardened and tempered the same as engine lathe tools (see *Elements of Machine Work*), and generally used in a speed lathe. See § 23.

325. Cutting speeds and angles for hand tools. — The cutting speed for steel, wrought iron and cast iron is greater than that for engine lathe cutting tools,— from 150 F.P.M. to 225 F.P.M., — as the cuts are lighter and not so continuous. See Hand Tools for Brass Finishing, § 341. The cutting angle and clearance are obtained both by grinding and by the manner in which tool is presented to work, the latter depending somewhat on the height of Tee rest.

326. Methods of holding work in speed lathe depend on the shape of the work: first, by centering and mounting work upon the centers; second, by holding in chuck; third, by holding one end in chuck and centering projecting end, using dead center to steady work. Small work of curved outline, except chuck work, can be more rapidly finished in a speed lathe.

All metals are hand turned without a lubricant.

327. The Tee rest is used to support the tool, and should be placed from $\frac{1}{8}''$ to $\frac{3}{8}''$ from work, depending on kind of tool used. With a few exceptions, the cutting edge of tool should be the same height as the lathe centers.

328. The graver, Fig. 216, except for inside rounds, is the most useful hand turning tool for both roughing and finishing.

Digitized by Google

The cutting edges are AB and $A'B$. They are obtained by grinding $B'C$ to an angle of about 50° with the body of the tool.

Attention. — On account of the keen cutting angle of the graver, care should be taken when using it on brass. See § 341.

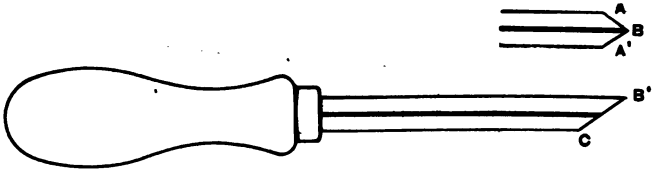


FIG. 216. — GRAVER OR DIAMOND-POINT HAND TOOL.

329. To rough turn cylindrical surfaces with graver, Fig. 217. — Work D is mounted on centers. Graver F is held on the rest horizontally, to turn cast iron. For steel, wrought iron, or copper, the handle is lowered to give it rake. To bring tool

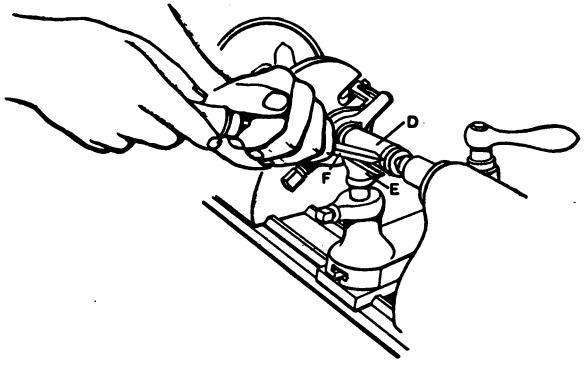


FIG. 217. — ROUGH TURNING WITH GRAVER IN SPEED LATHE.

to its cut, elevate handle. To rough turn work, pivot graver upon rest, move point from right to left parallel to work. Test work frequently with calipers. See Slide Rest, § 352.

330. To finish turn cylindrical work with graver, Fig. 218.— Press graver G hard on rest H with edge nearly parallel to

Digitized by Google

work so that it will drag, carry finishing cut toward headstock, or vice versa.

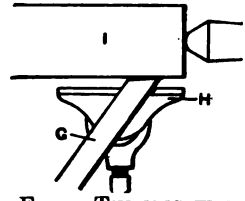


FIG. 218. — FINISH TURNING WITH GRAVER.

331. Step method of squaring with graver, Fig. 219.— A number of short cuts are taken on work *J* with graver *K* until the center is reached. In Fig. 220 the graver is turned over and the cut started at center and carried outward by depressing handle to finish square.

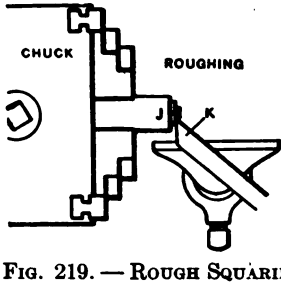


FIG. 219. — ROUGH SQUARING WITH GRAVER BY STEP METHOD.

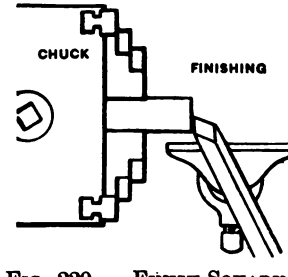


FIG. 220. — FINISH SQUARING WITH GRAVER.

332. The round-nose hand tool, Fig. 221, is used for inside rounds and fillets, and can be used for rough turning cylin-

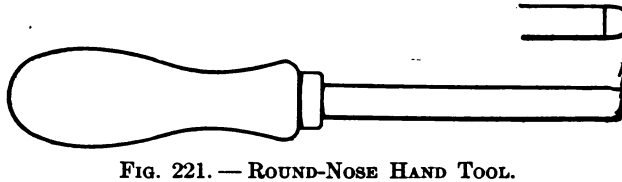


FIG. 221. — ROUND-NOSE HAND TOOL.

drical work. When employed on steel or wrought iron, it will cut better if top face is ground hollow, as at *A*, to give it rake. It is used in a manner similar to the graver.

Digitized by Google

MAKING MACHINE HANDLES.

333. Single handles are made as outlined in Figs. 222, 223, and 224. When manufactured in quantities, they are drop-forged to shape or made from bar stock with a forming tool in

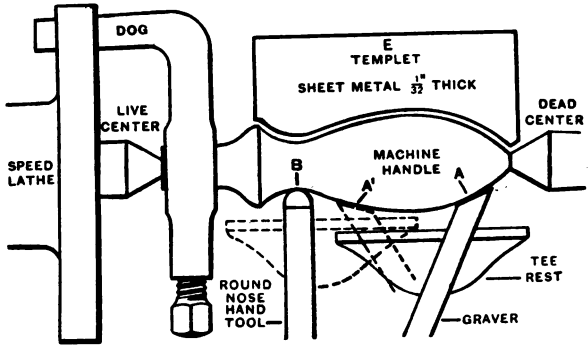


FIG. 222. — CURVED TURNING WITH HAND TOOLS. TESTING WITH TEMPLET.

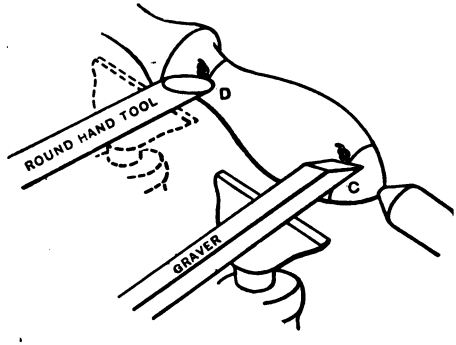


FIG. 223. — FINISHING MACHINE HANDLE WITH HAND TOOLS. OVERHEAD TURNING.

a turret lathe. By either of the latter methods, they require but a small amount of hand finishing.

334. Templets of sheet brass or steel, *E*, Fig. 222, are used for duplicate work, serving as a pattern or guide to uniform production.

Digitized by Google

They are lined out from specifications on the drawing, and are rough cut by chipping, or with shears, and finished by filing. Templets are used as guides when rough turning or planing, and for careful tests when finishing.

335. To make a machine handle, Fig. 224.

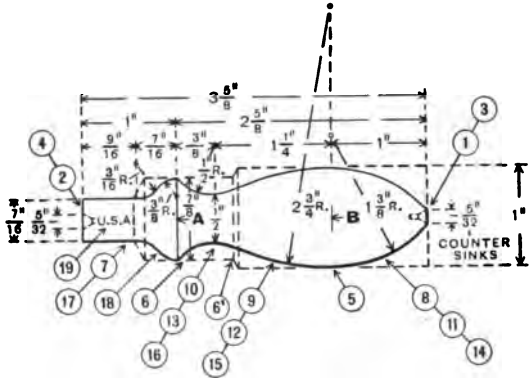


FIG. 224. — SCHEDULE DRAWING OF MACHINE HANDLE.

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.

ROUGHING. FINISHING.

Stock, machine steel $\frac{1}{16}''$ to $\frac{1}{8}''$ large.

True live center. Set dead center in approximate alinement.

Lard oil may or may not be used in squaring and turning steel or wrought iron.

Use carbon-steel cutting tools. See *Exception*, p. 59.

OPERATIONS.	MACHINES, SPEEDS, FEEDS.	TOOLS.
File ends flat.	Vise.	File.
Center.	Centering machine. Drill, 1250 R.P.M. Countersink, 700 R.P.M.	$\frac{3}{32}''$ drill, 60° countersink, lard oil.
Rough square, (1), (2). See Step Method of Rough Squaring, § 178.	Engine lathe, 12 to 16″. 2d speed, or 35 F.P.M. Hand feed.	Dog, side tool, 35° rake, calipers, rule.
Recenter.	Speed lathe, 2d or 3d speed.	

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.

Continued.

Operations.	Machines, Speeds, Feeds.	Tools.
Finish square, (3), (4).	Engine lathe, 3d speed, or 50 F.P.M. Hand feed.	
Rough turn $\frac{1}{64}''$ large, (5). One cut. Turn one-half of length, reverse and turn the other half.	2d speed, or 35 F.P.M. Medium power feed — 80 to 1″.	Diamond-point tool, or holder and cutter, 35° rake, calipers, rule.
Rough turn $\frac{1}{64}''$ large, (6). One cut. Turn to (6′), about $1\frac{1}{2}''$ from end.		
Hold in vise and mark lines 1″ from each end at (A) and (B).	Vise.	Copper sulphate, scriber, rule.
Rough turn $\frac{1}{64}''$ large, (7). Two or three cuts.	3d speed, or 35 F.P.M. Medium power feed — 80 to 1″.	Calipers, rule.
Rough turn convex portions, (8), (9), (10), several cuts. Operate both feeds in combination, by hand, and test with templet, as at *E*, Fig. 222.	3d speed, or 50 F.P.M.	Copper under set screw of dog, diamond-point tool, or holder and cutter, 35° rake, templet.
Rough turn with graver, (11), (12). See *A* and *A′*, Fig. 222. Test often with templet.	Speed lathe, 8 to 12″, 2d speed, or 150 or F.P.M.	Graver.
Rough turn, (13). See *B*, Fig. 222. Test with templet and calipers at diameters $\frac{1}{2}''$ and 1″.	.....	Round-nose hand tool, templet, calipers, rule.
Finish turn, (14), (15). See *C*, Fig. 223.	3d speed, or 225 F.P.M.	Graver, templet, calipers, rule.
Finish turn, (16). See *D*, Fig. 223.	.....	Round-hand tool, templet, calipers, rule.
Finish turn, (17).	Engine lathe, 12 to 16″, 3d speed, or 50 F.P.M. Fine power feed — 140 to 1″.	Diamond-point tool, or holder and cutter, 35° rake, calipers, rule.

Digitized by Google

SCHEDULE OF MACHINES, OPERATIONS AND TOOLS.

Concluded.

Operations.	Machines, Speeds, Feeds.	Tools.
Rough and finish turn, (18). Copy the reverse curve (the ogee) as accurately as can be determined by the eye, or make a templet from specifications or curve on drawing. See § 334.	Speed lathe, 3d speed, or 225 F.P.M.	Graver, round-nose hand tool.
File, (14) and (15).	3d speed, or 175 F.P.M.	8″ to 10″ mill bastard file.
File, (16) and (18).....	.....	5″ to 8″ half-round smooth file.
Polish, (14), (15), (16), (18).	Highest speed.	60, 90, 120 and flour emery cloth and stick, oil.
Stamp name or initials, (19).	Vise.	Steel name stamp, hammer, copper jaws.

Attention. — Part 17 is usually fitted to a hand wheel, or other machine part, by a forcing or threaded fit; if so, operation 17 is omitted to allow the desired fit to be made later.

Note. — The machine handle may be finished with graver and round-nose hand tool used horizontally, as in Fig. 222, or with graver and round hand tool, by overhead turning, as in Fig. 223.

DRILLING, TAPPING, AND HAND THREADING IN SPEED LATHE.

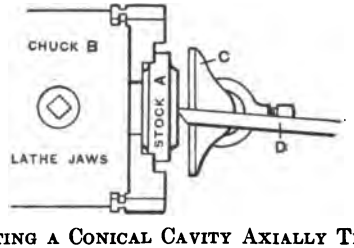


Fig. 225. — Cutting a Conical Cavity Axially True with Graver to Start Twist Drill.

Digitized by Google

336. To cut a conical cavity axially true with graver, Fig. 225. — Hold nut *A* in chuck *B*. Set Tee rest *C* to bring edge of graver *D* at height of center. Hold graver down hard on Tee rest and at the same time press it into work. If graver is not held firmly, it will not produce a cavity axially true. If the work is brass, it is best to use the corner of the planisher. See Planisher, § 344.

337. To chuck with twist drill in speed lathe, Fig. 226.— Make a conical cavity in work *E*, true to axis of rotation and, approximately, to the angle and diameter of drill. Fasten dog *G* to drill *F* and place in conical hole, with dog on rest *J* and dead center *H* in end of drill. Grip dog with left hand, start lathe and feed drill toward headstock with the right hand.

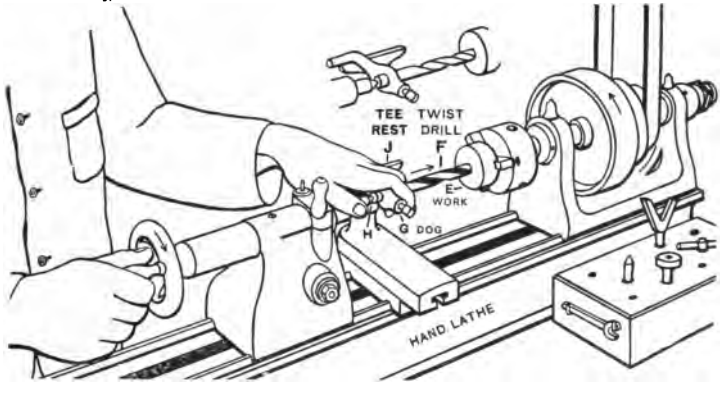


FIG. 226. — CHUCKING WITH TWIST DRILL IN SPEED LATHE.

Attention. — Hold drill firmly and do not allow it to draw off dead center, which may cause drill to break and may spoil the work.

338. To tap work in speed lathe, Fig. 227. See § 534. — Place tap wrench *K* on tap *L* and then insert in hole in work *M*, held in chuck *N*. Guide tap by dead center *O*, allowing handle of wrench to bear on rest *P*.

Digitized by Google

Rotate work by pulling belt Q, or rotate cone R, and feed footstock spindle to follow up tap with dead center.

Attention. — Carefully follow up tap with dead center, as the tap will break if it slips off.

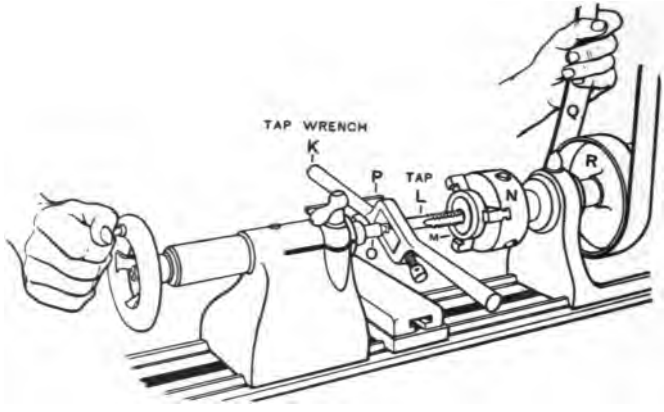


FIG. 227. — TAPPING IN A SPEED LATHE.

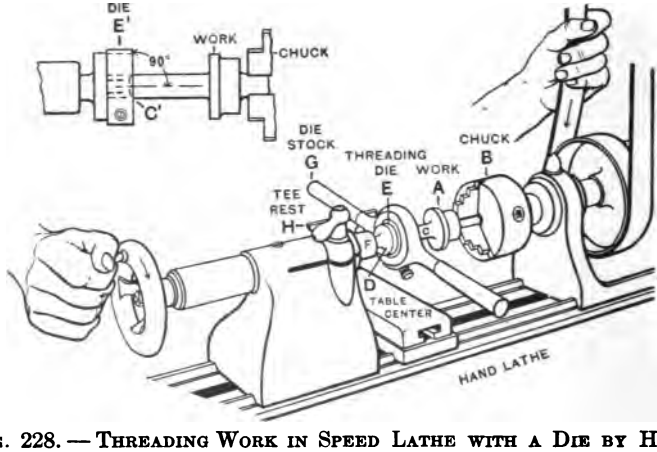


FIG. 228. — THREADING WORK IN SPEED LATHE WITH A DIE BY HAND.

As small taps break easily, it is best only to start tap in the lathe and then finish at vise.

339. To thread work held in chuck with die and stock by hand, Fig. 228. — Chamfer end of work A, held in chuck B, as

Digitized by Google

at C', to assist starting die centrally. Remove dead center, insert table center D, place muzzle side of die E against work (see E') and handle G on Tee rest H. Feed spindle F in until D presses against die stock. Rotate lathe by hand, following up die with D until thread is started. To terminate thread abruptly reverse die.

Attention. — The work must be adjusted in chuck to run true, otherwise die will cut a crooked thread.

340. Hand chasers are used to cut fine threads (20 to 70 P) on brass tubing, and may be used to finish threads partly cut. The teeth of chasers are cut by milling or by pressing the blank against a master or hob tap revolved in a lathe.

Screws, Fig. 229, are threaded as indicated at A, B and A', B'. Run lathe at medium slow speed. Carefully move chaser along rest one pitch per revolution of work. After the thread is started it helps to guide the chaser. Lack of experience or carelessness may cause the chaser to produce a double thread or a drunken thread (irregular spiral).

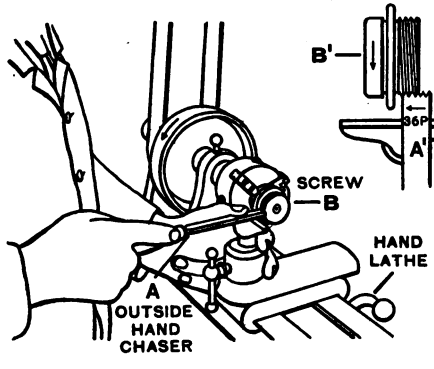


FIG. 229. — CHASING OUTSIDE THREADS.

Nuts, Fig. 230, are threaded as indicated at A, B and A', B'.

Attention. — To thread successfully with hand chasers requires intelligent practice. They are not used as much as formerly, except on thin tubing, on account of the improvement in taps and dies.

Digitized by Google

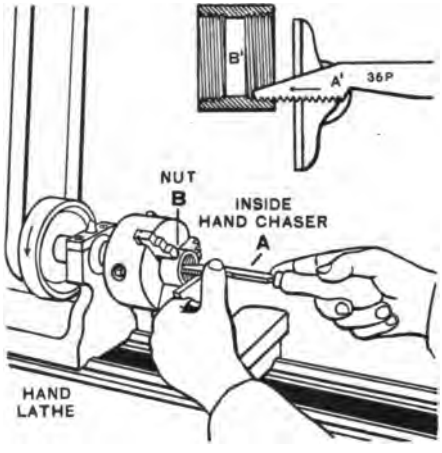


FIG. 230. — CHASING INSIDE THREADS.

341. Hand tools for brass finishing. — Brass and composition castings may be hand turned at a speed of 500 F.P.M. Bronze or gun metal, which is harder than brass, may require a slower speed. Rolled brass may be turned at a speed nearly as high as hard wood. See Brass Turning in Engine Lathe, § 357.

342. Ripper or rough turning tool, Fig. 231. — The top and side views are shown at *A*, *B*, and end view at *C* and *C'*,

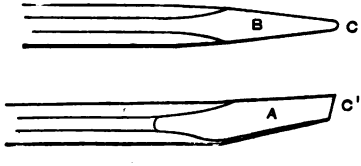


FIG. 231. — RIPPER OR ROUGH TURNING TOOL FOR BRASS.

The ripper is shaped by forging, and sharpened by grinding and oilstoning. It is a good tool to rough turn brass.

343. Method of using ripper, Fig. 232. — Hold tool as shown at *D*, *F* and *G*. Place tee rest close to work *H*, use

Digitized by Google

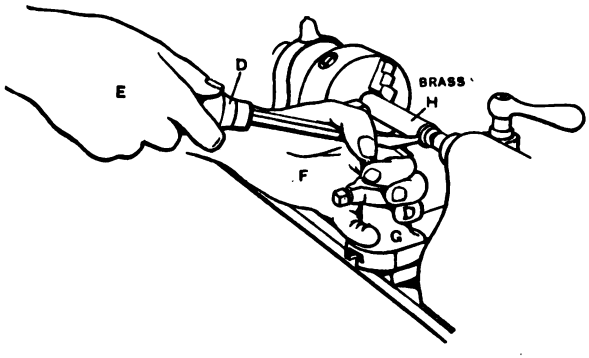


FIG. 232. — ROUGH TURNING A BRASS CASTING WITH RIPPER.

it as a fulcrum, and move handle from right to left, move tool along rest and repeat operation. Rough turn work $\frac{1}{32}''$ large, and finish turn with planisher.

344. Planisher or flat scraper. — Fig. 233 (*A*, *B*) is used for finishing brass. The cutting end, *C*, *C'*, is forged, and ground and oilstoned on end, sides and edges.

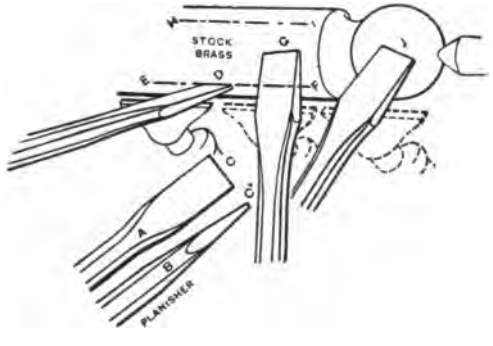


FIG. 233. — FINISH TURNING BRASS WITH PLANISHER.

345. Method of using planisher, Fig. 233. — Hold similarly to ripper. Place at angle indicated at *D*, slide along line *EF* from right to left or vice versa. If it chatters, use it at top of

Digitized by Google

work and reduce length of cutting edge, as at *G*, by carrying cutting edge at slight angle with work, as indicated by line *HI*.

To finish a spherical surface, hold planisher as at *J*.

Fig. 234, *K*, *L*, *M*, and *N*, shows a few of the purposes, both roughing and finishing, for which a planisher may be used in its regular or special form.

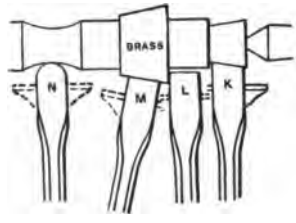


FIG. 234. — ROUGHING AND FINISHING ROLLED BRASS WITH PLANISHERS.

346. Filing brass in the lathe. — It is usually unnecessary to remove tool marks preparatory to polishing by filing as brass can be scraped smooth with a planisher.

347. Burnishers and burnishing metal. — After polishing and buffing, plated or unplated work may be given a brilliant luster by using a steel burnisher, shown at *A*, *B*, Fig. 235.

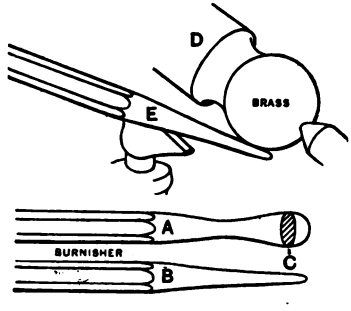


FIG. 235. — BURNISHING TO PRODUCE HIGH FINISH.

The burnisher is elliptical in cross-section, as at *C*, is made glass hard, highly polished, and by skillful application will impart a mirror-like surface.

Digitized by Google

Work *D* is revolved at high speed and the burnisher is lubricated with vaseline, kerosene, or a solution of borax and water and is moved along the work slowly with sufficient pressure to produce the desired luster.

Jewelry, cutlery, silverware, etc., are burnished by hand with hardened steel tools and also with tools faced with agate or bloodstone.

348. Hand cutting-off tools are forged and ground as in Fig. 236, *A*, *B* for brass and *C*, *D* for steel and wrought iron, and are used in hand lathe to cut off small stock, as follows:

Press lever *E*, Fig. 237, with the left knee to release spring chuck *F*, and insert rod *G* through spindle (or rod may be

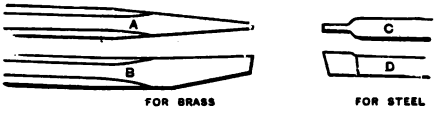


FIG. 236. — HAND CUTTING-OFF TOOLS.

held in universal chuck *H*). Hold tool *J* on rest *K* at height of center, and force inward. To prevent brass cutting-off tool from binding, cut sideways to increase width of groove. See Lathe Cutting-off Tools, § 197.

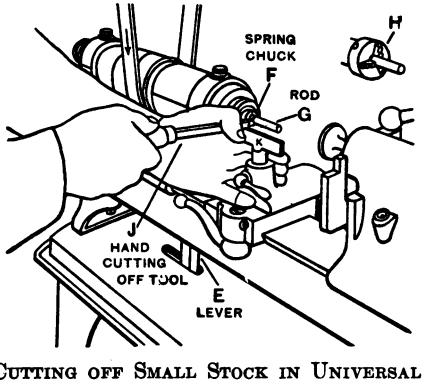


FIG. 237. — CUTTING OFF SMALL STOCK IN UNIVERSAL HAND LATHE.

349. Improved hand cutting-off tool. — After adjusting the tool on the rod the edge of circular cutter *A*, Fig. 238,

Digitized by Google

is forced into the $\frac{1}{2}''$ rod by lightly pressing the handles together. Stop B can be set to cut off equal lengths. Bushings are used in holder C for smaller rods.

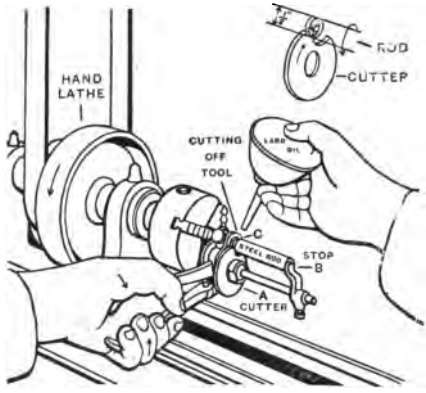


FIG. 238. — IMPROVED HAND CUTTING-OFF TOOL.

350. Inside hand turning or boring. — Work A, Fig. 239, is held in chuck B. Tee rest C is placed close to work and

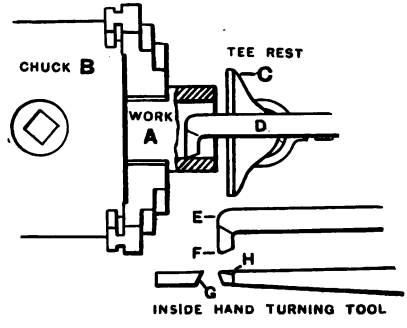


FIG. 239.— HAND BORING AND INSIDE TURNING.

adjusted to bring top of tool D to center of work. Tool D is then fed inward parallel to axis of work. Point F is ground with some clearance, as at G. For steel or wrought iron, it is given rake, as at H.

351. Hand tools from old files. — In an emergency, hand tools of different shapes can be made from old files by first

Digitized by Google

grinding off the teeth, then grinding or forging to desired shape.

352. A slide rest, Fig. 240, is a very convenient attachment for a speed lathe for squaring and turning work either mounted on centers or held in a chuck.

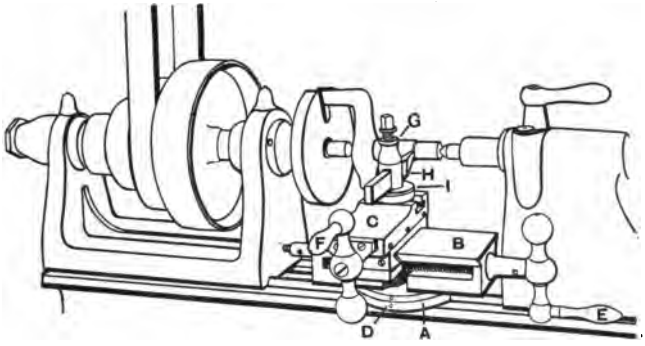


FIG. 240. — TURNING IN SPEED LATHE WITH SLIDE REST.

SCHEDULE OF PARTS.

A — Base fastened to tee rest slide.	*F* — Handle, cross feed.
B — Long slide, pivoted on *A*.	*G* — Tool-post.
C — Cross slide.	*H* — Cutting tool.
D — Zero line, for taper turning.	*I* — Concave and curve washers for raising and lowering point of tool.
E — Handle, long. feed.	

SPRING WINDING.

353. Coil springs, both extension, Fig. 241, and compression, Fig. 242, made of wire $\frac{1}{16}$″ and less in diameter, are wound in a speed lathe; larger sizes are best wound in an engine lathe with back gears in. For sizes of wire see §§ 659–661, 663.

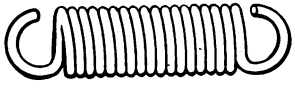


FIG. 241. — EXTENSION SPRING.

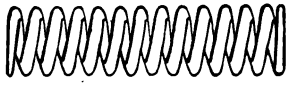


FIG. 242. — COMPRESSION SPRING.

Digitized by Google

354. Spring winder, *A*, Fig. 243, for winding compression or extension springs. To wind compression spring mandrel *C*, rotate web on spacing wheel *D* into position, equal in thickness to space between coils. Pass wire *E* through lug *F*, between friction washers through stud *G* and through hole in mandrel to fasten. Press spacing wheel against mandrel and adjust nut *H* to give desired tension. Rotate lathe by hand or power, and wind spring. To make each end flat, wind first and last coils straight.

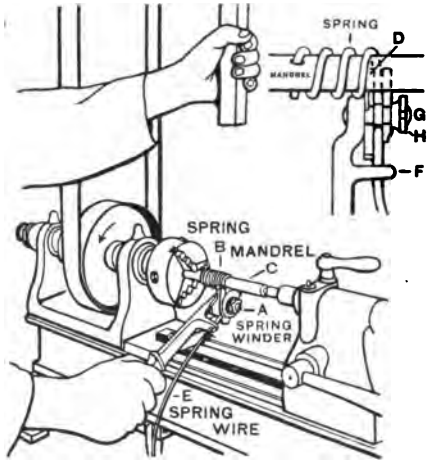


FIG. 243. — WINDING A COMPRESSION SPRING.

Extension springs are wound in the same manner, except that the thickest web of the spacing wheel is pressed against the spring after one turn to keep it closed.

Springs when released by cutting off the wire at both ends, expand some by uncoiling. If the spring is brass, this may be prevented, to some extent, by blows of a lead hammer applied over the whole circumference; but if the spring is made of piano wire, a smaller mandrel should be used to allow for the expansion. Large steel springs are prevented from expanding by heating red hot before releasing, and hardening and tempering afterwards.

Digitized by Google

This spring winder may also be used to wind springs by around a mandrel held in a vise. Machines, both hand and automatic, are obtainable for winding springs.

355. To gear a lathe to wind coil springs. — Arrange as for screw cutting. For a compression spring, the pitch should equal the space desired between the coils plus diameter of wire, and for an extension spring the pitch should equal diameter of wire. Mount mandrel on centers with dog or hold end in chuck. Pass wire between two blocks in tool-post to give desired tension, fasten end of wire, start lathe and wind spring.

356. To wind a compression spring with a wire hook. — Place hook between the coils and have assistant at the back of the lathe hold end or allow it to hang between the ways and the carriage. Feed carriage along slowly by hand with a piece of steel in the tool-post pressing against the spring.

Warning. — Be careful in releasing a spring from its winding arbor by using up all the wire or by cutting off, as it usually uncoils a little, suddenly, and with a force that may injure the hands.

Digitized by Google

CHAPTER XIII.

BRASS FINISHING. MACHINING BRONZE, COPPER, ETC. NURLING. CURVE TURNING AND FORMING. STEADY AND FOLLOWER RESTS.

BRASS FINISHING.

357. To turn brass in the engine lathe. — Brass, also known as composition, ranging in hardness from soft yellow to hard brass or bronze, is generally turned with tools having little or no rake. High speed and quick feeds are used and the tools must be kept sharp by frequent grinding to obtain the best results. See To Machine Bronze, § 362. For Cutting Speed, see § 98.

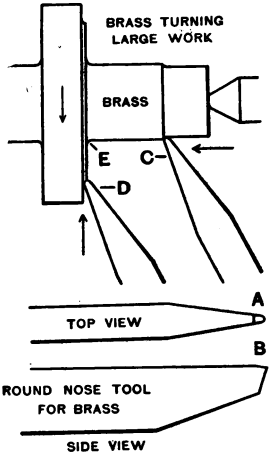


FIG. 244. — BRASS TURNING WITH ROUND-NOSE TOOL.

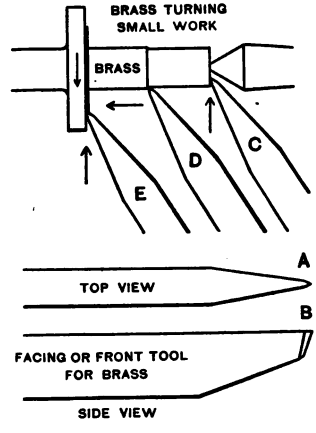


FIG. 245. — BRASS TURNING WITH FRONT TOOL.

358. The round-nose tool, as at *A* and *B*, Fig. 244, is generally used for large brass work. It may be used to turn the

Digitized by Google

diameter as at *C*, the shoulder, as at *D*, and the round corner or fillet as at *E*, without change of position in the tool post.

359. Use front tool, Fig. 245, with point *A*, *B* ground at angle from 50° to 60° for squaring, turning, or facing, without change, as at *C*, *D*, and *E*. If the work is rigid, cut inward or outward; if slender, cut inward only. This tool may be ground to fit thread gage, and used as a threading tool for United States Standard or Sharp V threads.

Attention. — As bronze and brass will stretch, it is necessary, when fitting a steel screw to a bronze or brass nut, to force the screw in hard at first to avoid a loose fit.

Reamed and drilled holes in brass or bronze are usually smaller than holes made with the same tools in cast iron on account of the resilient and adhesive quality of these materials.

Taps, dies, reamers, and files that are used on steel or iron may be used on brass, but it is best to have separate tools, for such tools never work as well on brass after they are used on steel or iron.

Brass in general is machined dry, but to thread rolled brass, as rod and tubing, with a die, it is usually necessary to use lard oil to prevent the chips from clinging to the threads of the die and stripping the threads on the work.

360. To make brass binding post, Fig. 246. —

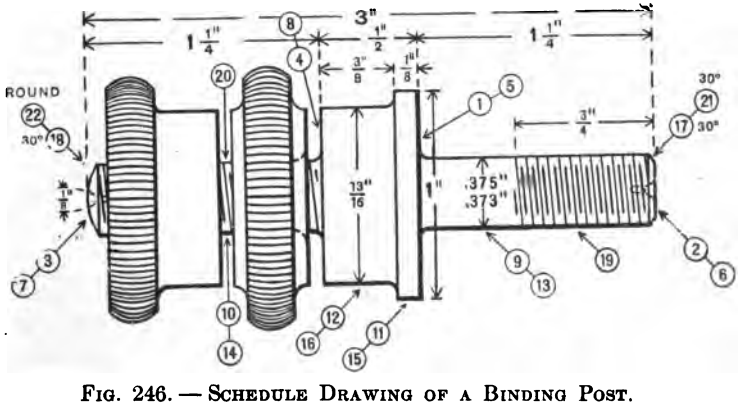


FIG. 246. — SCHEDULE DRAWING OF A BINDING POST.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.

ROUGHING. FINISHING.

Stock, composition or brass castings $\frac{1}{16}''$ large.

True live center. Set dead center in approximate alinement.

Use carbon-steel cutting tools. See *Exception*, p. 59.

OPERATION.	MACHINES, SPEEDS, FEEDS.	TOOLS.
File ends square and center to $\frac{1}{8}''$ diameter.	Vise, centering machine, drill, 3500 R.P.M., countersink, 1700 R.P.M.	File, $\frac{1}{16}''$ drill, 60° countersink.
Rough square, (1), (2), (3), (4). Take as little as possible off 1. See §§ 358, 359.	Engine lathe, 12″ to 16″, 4th or 5th speed, or 90 F.P.M. Hand feed.	Round-nose tool $\frac{1}{8}''$ wide, (1), (4), front tool, (2), (3), calipers, rule.
Recenter to $\frac{1}{8}''$ diameter.	Speed lathe, 3d or 4th speed.	60° countersink.
Finish square, (5), (6), (7), (8).	Engine lathe, 4th or 5th speed, or 150 F.P.M. Hand feed.	Round-nose tool, (5), (8), front tool, (6), (7), calipers, rule.
Rough turn diameter to $\frac{1}{64}''$ large, (9), (10), (11), (12).	4th or 5th speed, or 90 F.P.M. Medium power feed — 80 to 1″.	Copper under set screw of dog, round-nose tool, calipers, rule.
Finish turn, (13), (14), to fit $\frac{3}{8}''$ gage, or use micrometer, easy fit, tool finish, filing not necessary. Finish turn, (15), (16.)	4th or 5th speed, or 150 F.P.M. Fine power feed — 140 to 1″.	Calipers, rule, $\frac{3}{8}''$ mandrel, $\frac{3}{8}''$ ring gage, or micrometer.
Chamfer ends 30° to depth of thread, (17), (18).	Hand feed.	Front tool.
Thread in engine lathe to fit nuts, (19), (20). See § 226, or	2d or 3d speed, or 25 F.P.M. Arrange change gears for 24 threads.	U.S.F.-threading tool.
with $\frac{3}{8}''$ × 24 U.S.F. die, 20). Start in lathe. lt by hand, and finish . See § 339. If de- ost and nuts may be ed $\frac{3}{8}''$ × 16, United , Standard thread.	Speed lathe — 8″ to 12″. Vise.	3″ universal chuck, $\frac{3}{8}''$ × 24 U.S.F. thread die and die stock, special vise jaws. See Automobile Screws, §§ 639,640.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.
Concluded.

Operations.	Machines, Speeds, Feeds.	Tools.
Rechamfer, (21).	Engine lathe, 4th or 5th speed, or 90 F.P.M.	Clamp nut, front tool.
Round end with forming tool, or	.....	Forming tool.
By hand tool, (22).	Speed lathe, 4th or 5th speed, or 500 F.P.M.	Planisher or graver.

361. To make brass nurled thumb nuts, Fig. 247.

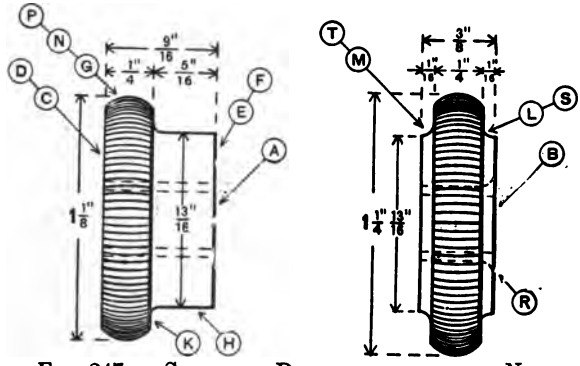


Fig. 247. — Schedule Drawing of nurled Nuts.

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.

Operations.	Machines, Speeds, Feeds.	Tools.
Chuck, drill, and tap nuts, (A), (B), §§ 336–338.	Speed lathe, drill, 680 R.P.M. Vise.	3″ universal chuck, planisher or graver, P or $\frac{21}{64}$″ drill, $\frac{3}{8}$″ × 24 U.S.F.-thread tap, tap wrench.
Screw on nut mandrel, mount on centers, rough square, (C). Finish square, (D).	Engine lathe, 4th or 5th speed, or 90 to 150 F.P.M. Hand feed.	Dog $\frac{3}{8}$″ × 24 U.S.F., nut mandrel, front tool.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.
Concluded.

Operations.	Machines, Speeds, Feeds.	Tools.
Reverse on mandrel, rough square, (E). Finish square to thickness, (F).	.....	Front tool, calipers, rule.
Rough turn to diameter, $1\frac{1}{8}''$, (G).	Engine lathe, 4th or 5th speed, or 90 to 150 F.P.M. Fine power feed — 140 to 1″.	Round-nose tool, calipers, rule.
Rough and finish turn to diameter, (H). Rough and finish square to thickness, (K).	.....	Round-nose tool, calipers, rule.
Square and turn nut (B) in same order as (A), except leave diameter (L) (M) $\frac{1}{64}''$ large.		
Round nut (N) to fit concave single nurl. See § 369.	Speed lathe, 4th or 5th speed, or 500 F.P.M.	Planisher.
Nurl (P), also round and nurl nut (B). See § 369.	2d or 3d speed or 300 F.P.M.	Hand nurling tool.
Hold nut mandrel in chuck, screw on nut (B), and recess (R) to fit against 4 on binding post, Fig. 246.	4th or 5th speed, or 500 F.P.M.	Chuck, planisher or graver, nut mandrel.
Screw nuts on binding post, mount on centers and hand turn, (S), (T), to correct diameter of post and nut (A).	4th or 5th speed, or 500 F.P.M.	Clamp nut, round-nose hand tool.
File.	3d or 4th speed, or 250 F.P.M.	5″ half-round smooth file.
Stamp name or initials at end, (C).	Place on metal block.	Steel name stamp, hammer.
Polish and lacquer posts and nuts all over, except threads.	Highest speed, or 6000 F.P.M. See §§ 321–323.	120 and flour emery cloth, crocus cloth (an oxide of iron), rouge (a red iron peroxide), lacquer, brush.

Digitized by Google

Attention. — The cutting speeds in schedule are for soft brass or composition castings; but if castings are hard, reduce the cutting speed.

MACHINING BRONZE, COPPER, ETC.

362. To machine bronze. — Phosphor, Tobin, and other bronzes are tougher than ordinary brass, and are machined with cutting tools similar to those used for steel and wrought iron, and lubricated with lard oil.

363. To machine copper. — Copper is machined with tools similar to those used for steel but preferably with more rake as a keen edge is desirable, and with the point slightly rounded. Lubricate with milk, soda water, or soap mixture. Use speed nearly as fast as for brass.

364. To machine aluminium. — Aluminium, owing to its light and ductile nature, is machined with tools having acute cutting angles, more rake than is used for steel, and at a moderately fast cutting speed. See Lubricants for Cutting Tools, § 100. A very high polish may be obtained on a cotton buffing wheel.

365. To machine Babbitt and lead. — Babbitt and lead are machined dry and with keen tools, the same as aluminium.

366. To machine vulcanite or hard rubber or fiber. — Rubber and fiber are machined dry, at a moderately fast speed, with cutting tools similar to those used for steel. They are finished and polished the same as steel, for a very high-finish buff on a cotton buffing wheel charged with tripoli or rottenstone.

367. To machine rawhide, use tools similar to those used for steel, but preferably with more rake and with a cutting speed about the same as for brass, and machine dry. For gear blanks, the several layers are confined between riveted plates, and should be shellacked as soon as machined to prevent swelling. Rawhide is milled and drilled dry, and the chips are removed by compressed air or a fan.

Digitized by Google

NURLING.

368. Hand and machine nurling or milling tools, Figs. 248 and 249, are used to check or mill surfaces of nuts, screw heads, handles, knobs, etc., to increase the grip and facilitate rotation. These indentations are similar to those on the edges of silver and gold coins to detect the removal of metal and called milled edges.

369. Hand nurling. Fig. 248. — Thumb nuts and screw heads are often nurled with single nurl wheel *A* and holder *B*, Fig. 248. Oil work and nurl.

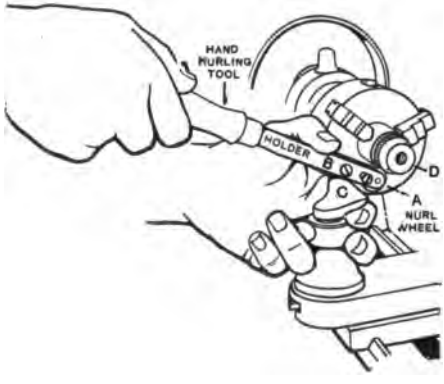


FIG. 248. — HAND NURLING IN SPEED LATHE.

SCHEDULE OF OPERATIONS.

Speed for hand nurling brass, 300 F.P.M.	Speed for hand nurling steel, 200 F.P.M.
Place holder on rest *C* which should be firmly clamped, and place wheel under thumb nut *D*. Press down firmly with right hand, at same time steadying tool with the left, until desired effect is produced. *Attention.* — Use fine nurls on steel.	

Digitized by Google

370. Machine nurling, Fig. 249.

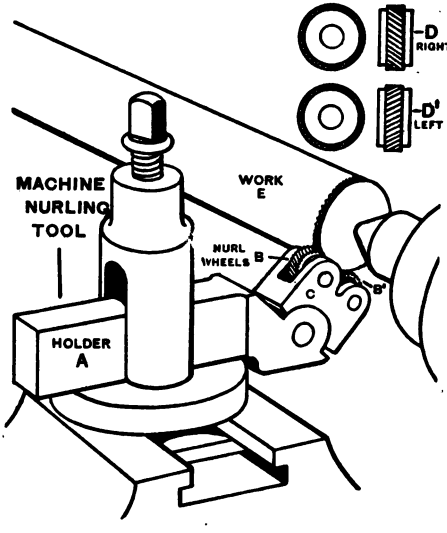


FIG. 249. — MACHINE NURLING IN ENGINE LATHE.

SCHEDULE OF OPERATIONS.

Speed for machine nurling iron and steel, 40 F.P.M.

Feed for nurling, medium power feed, 80 to 1″.

Nurling tool, Fig. 249, consists of holder *A* and two hardened steel nurling wheels *B*, *B'*, mounted in head *C*, connected with holder by rocking joint. Nurls have opposed helical teeth as at *D*, *D'*. Holder is fastened at right angles to work *E*, and adjusted to have both wheels bear equally on work.

Use power feed. Oil work and nurls. Start lathe. Force nurls with cross feed screw hard against work, preferably with half width of nurls. Stop lathe, and if pitch of checking on work is same as nurls, reduce pressure and throw in power long. feed to travel back and forth over surface until projections come nearly to a point.

Attention. — To prevent hand or machine nurls from cutting double set of projections twice as fine as the nurls and spoiling the work, press nurls hard at start until desired effect is produced, then partially relieve pressure before starting power feed, to avoid undue wear on centers and center holes.

Note. — Nurls for hand work are obtainable in a variety of patterns; nurls for machine work, in fine, medium, and coarse pitches. Medium is most used.

Digitized by Google

CURVE TURNING AND FORMING.

371. Curve turning. — Small outside and inside rounds (convex and concave surfaces), ogees and other irregular curves may be rough formed, as in Fig. 250, with one hand operating the cross feed and the other the long. feed. The tool

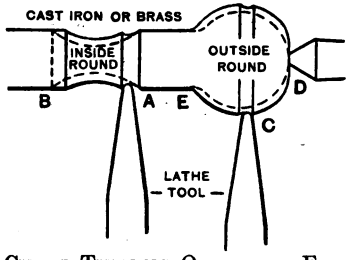


FIG. 250. — CURVE TURNING, OPERATING FEEDS BY HAND.

for the inside round is moved, approximately to the correct curve, repeatedly from *A* to *B*; for the outside round, from *C* to *D* and from *C* to *E*. By moving one feed more rapidly than the other, as the curve may require, good results may be obtained. If the work is slender or the curve large, it is usually finished with hand tools in the speed lathe (see Templet, § 333). If the curve is small and work stiff, forming tools may be used.

372. Forming tools for engine lathe work are used for forming duplicate pieces. They may be made by milling or filing. If much stock has to be removed, rough form with a

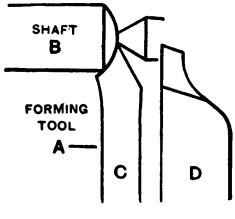


FIG. 251. — FORMING WITH FORGED TOOL.

suitable lathe tool. Forming tool *A*, Fig. 251, is rounding the end of shaft, after which it may be filed and polished.

Forming tools are also used in turret lathes, screw machines and planers.

Digitized by Google

373. Forming cutter and holder. — Forming cutters, to fit thread tool holders, are obtainable. In Fig. 252, forming cutter *A*, held in holder *B*, is forming wheel *C*.

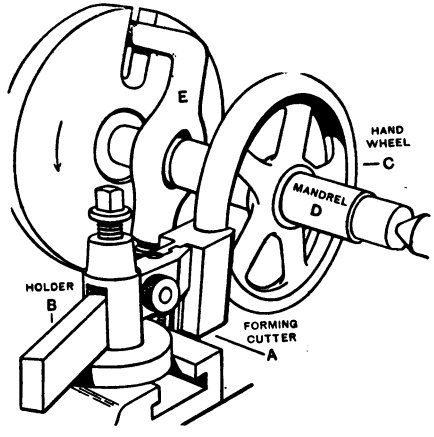


FIG. 252. — FORMING WITH HOLDER AND CUTTER.

The manufacturing method of machining hand wheel rims is to mill them with a vertical milling machine or form them in a special machine which carries a pointed tool mounted on a turn table around the curve.

STEADY AND FOLLOWER RESTS.

374. A steady rest, Fig. 253, is used to support a slender shaft to prevent vibration.

SCHEDULE OF PARTS.

A — Rest; two parts, — base and top.	*F* — Bolt, hinges top to base.
B — Base.	*G* — Clamp, fastens top to base.
C — Top, hinged to base.	*H* — Sliding jaws adjusted to spot *J* on shaft.
D — Clamp strap.	K_1 K_2 K_3 — Adjusting screws.
E — Bolt to clamp rest at any location, on ways of lathe.	*L* — Nuts to clamp jaws in position.

Digitized by Google

375. To turn spot on shaft, then adjust jaws to that spot and turn shaft, Fig. 253.

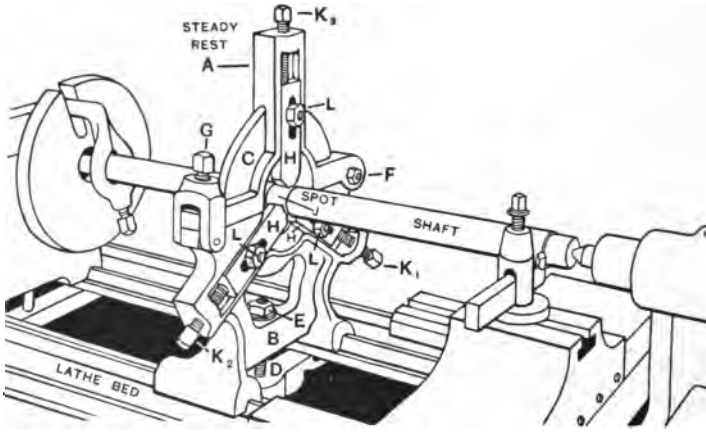


FIG. 253. — TURNING SLENDER SHAFT SUPPORTED BY A STEADY REST SET TO A TURNED SPOT.

SCHEDULE OF OPERATIONS.

True live center. Set dead center in accurate alinement.

1. Center and straighten shaft, rough square, recenter, finish square.

2. Turn spot J $\frac{1}{64}''$ to $\frac{1}{32}''$ large and wider than jaws H, central or nearer live center, and file smooth.

3. Move jaws H back to clear shaft and swing top C backward. Mount and clamp rest opposite spot. Adjust jaws H by screws K_1 first, then K_2 to touch shaft. Swing top C forward and clamp, and adjust third jaw H by screw K_3. Clamp jaws by nuts L and oil spot J.

4. Turn one-half shaft, reverse shaft, adjust jaws to turned portion and turn rest of shaft, or spot the shaft in center and rough and finish both halves to spot by reversing shaft, then move rest along toward live center, readjust jaws and finish spot to size.

376. The cat head, M, Fig. 254, is used to hold slender shafts to be spotted as well as for steadying slender shafts to

Digitized by Google

be turned without spotting. It is also used to hold square, hexagonal, or irregular section work to be turned, as valve stems that have one part square and the other round.

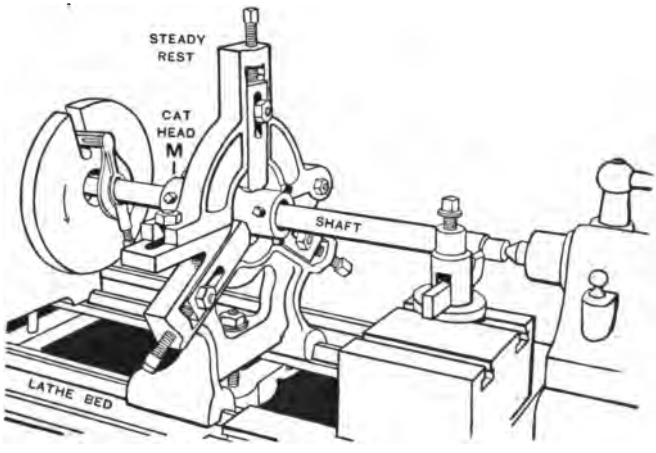


FIG. 254. — TURNING SLENDER SHAFT SUPPORTED BY A STEADY REST SET TO A CAT HEAD.

377. To true cat head on shaft. — Place it on shaft and true up accurately by adjusting screws. Test its truth by chalk, copper tool, or a test indicator. If the cat head is used in turning the second end of the shaft, it must be placed on the turned part and trued up as before. Usually the jaws of the steady rest are set directly on the shaft for the second half.

Attention. — Do not adjust jaws carelessly to cause the shaft to spring. If piece springs when outside skin is removed, straighten in a press, or if the shaft is slender, mount on centers and straighten. Use wooden jaws in steady rests for finished work.

378. Follower (Traveling) rest, *A*, Fig. 255, is used for turning shaft from end to end, and it is more convenient and produces more accurate work than a steady rest. The rest consists of two jaws and a frame bolted to carriage *B*. A spot of the desired diameter is turned at end *C* of shaft *D*, and the jaws

Digitized by Google

E and *E'* are adjusted to it. The tool must be slightly in advance of the jaws, which should be well lubricated where

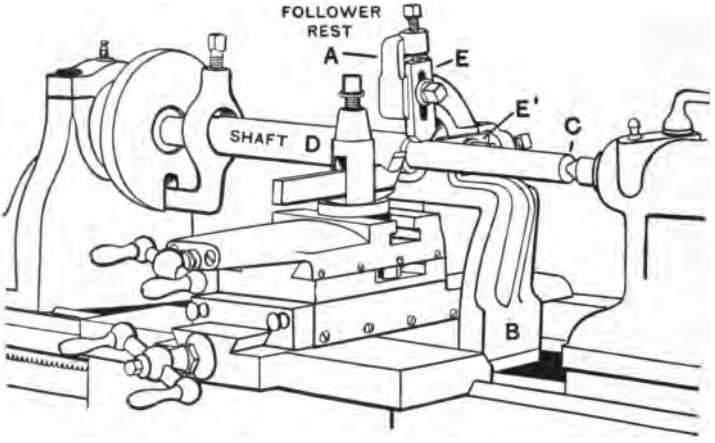


FIG. 255. — TURNING SLENDER SHAFT SUPPORTED BY A FOLLOWER REST.

they bear on the work. One-half of shaft is turned, then it is reversed and the second half turned. For more effective support, use bushings to suit different diameters of shafts in place of jaws. When cutting Square or 29° threads on slender pieces a follower rest is necessary to support the work.

Digitized by Google

CHAPTER XIV.

CYLINDRICAL GRINDING MACHINES. GRINDING WHEELS. PROBLEMS IN CYLINDRICAL GRINDING.

CYLINDRICAL GRINDING MACHINES.

379. Machine grinding is a scientific method of producing cylindrical, conical, and plane surfaces accurately, rapidly, and economically, with automatic grinding machinery; also for duplicating parts.

Almost any material, as hardened and soft steel, wrought iron, cast iron, brass, copper, aluminium, vulcanite, and wood fiber, may be ground accurately. Grinding machines are not designed to remove a large amount of stock but to produce accurate dimensions on work that has been roughturned.

The principle of grinding on two dead centers. — Ground work is more accurate than turned work. This accuracy is obtained by grinding the work on two dead centers as in

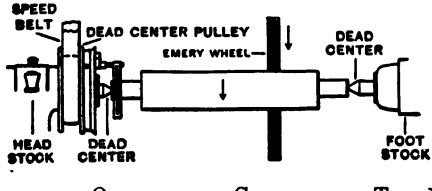


FIG. 256. — ACCURACY OBTAINED BY GRINDING ON TWO DEAD CENTERS.

Fig. 256, which eliminates the error caused by wear of spindle bearings. The wear of dead centers is readily corrected by grinding the centers, but the wear of spindle bearings cannot be readily corrected.

Machine grinding is also used for finishing as a substitute for polishing as it is more economical than using files and emery cloth.

Digitized by Google

380. Grinding machines may be divided into four general classes: Universal grinding machines for general work; plain grinding machines for outside work only; cutter grinding machines for grinding cutters and similar work; and surface grinding machines for surface grinding only.

Special grinding machines are designed for special purposes, as internal grinding machines, piston grinding machines, and crank grinding machines to rough as well as to finish automobile crank shafts without rough turning.

381. Universal grinding machine, Fig. 257, like the engine lathe, is designed for general work, as straight and taper grinding, both external and internal, facing sides of disks, grinding clearances on cutters, reamers, etc. See Internal, Cutter and Surface Grinding, *Advanced Machine Work.*

SCHEDULE OF PARTS.

COUNTERSHAFT DRIVE. BELT FEED.

382. *Machine Parts.*

A — Base, contains locker for small parts.

B — Bed, contains feed and reversing, mechanism.

C — Table.

C′ — Swivel table.

D — Swivel headstock (90°), position adjustable.

E and *E′* — Dead centers.

F and *F′*—Dead center pulleys; large or small may be used.

G — Live spindle pulley.

H — Stop for live spindle pulley.

J — Footstock, position adjustable.

K — Lever to spring dead center "in" or "out."

L — Grinding wheel, head mounted on swivel slide and adjustable "forward" or "backward;" it can also be fed by hand to grind steep tapers.

M — Grinding wheel.

N — Water supply pipe.

P — Belt to drive work.

Q — Belt to drive grinding wheel.

R — Belt to drive feed cone.

S — Belt to drive pump.

383. *Countershaft Parts.*

I. Line shaft.

II. Tight and loose pulley mechanism controlled by shipper.

III. Belt to drive sleeve carrying feed cone.

IV. Belt to drive work drum.

V. Work drum.

VI. Clutch operated by shipper brake.

VII. Belt to drive grinding wheel drum.

VIII. Grinding wheel drum.

IX. Belt to drive dead center pulley *F* to give higher speed.

X. Belt to drive live spindle pulley *G*.

Digitized by Google

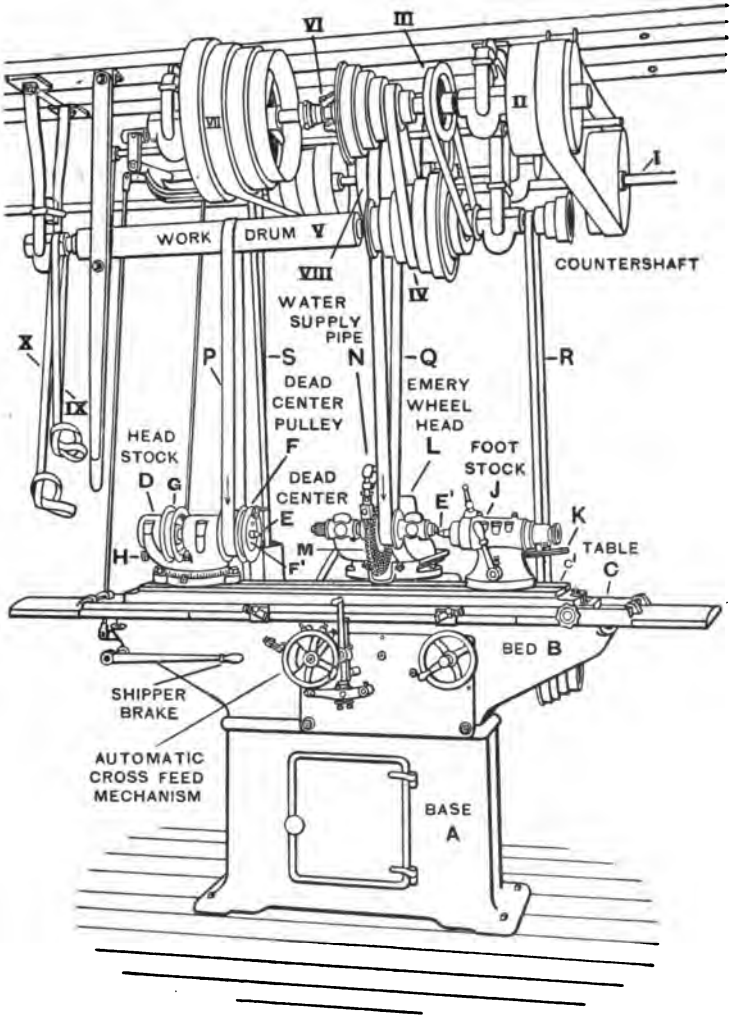


FIG. 257. — UNIVERSAL GRINDING MACHINE.

Digitized by Google

GRINDING WHEELS.

384. Grinding wheels, Chart, Fig. 258, are made from various abrasives, — emery, corundum, alundum, and carborundum. The abrasive is mixed with a bond, the amount and composition of which makes a wheel hard or soft and determines its grade or degree of hardness.

They are molded, baked, turned, balanced, and tested for hardness and speed.

The number of a wheel is determined by the number of meshes per linear inch of the sieve through which the abrasive has passed, and for grinding wheels runs from No. 12 to No. 120. Lower numbers indicate coarser wheels and higher numbers finer wheels. See *Elements of Machine Work.*

385. Coarseness. Grade of Hardness. Process.

Coarseness determined by No. of abrasive.

Hardness determined by bond binding grains of abrasive together.

Processes.

Vitrified.	Tanite.
Silicate.	Vulcanite.
Elastic.	Celluloid.

Each process is best adapted to certain *classes* of grinding, yet each may be used for *general* grinding.

386. Vitrified wheels, reddish brown in appearance, are made by mixing the abrasive with a bond of clay, sand, spar, etc., they are then molded into shape, placed in ovens similar to pottery kilns and subjected to a prolonged and intense heat (3000° F.). These wheels are even in texture, open and porous. They are especially adapted to grinding hardened steel, as they will not glaze easily and are thus cool-cutting. Wheels made by this process are used for general work and for cylindrical and cutter grinding. See Table of Grades, § 393.

Digitized by Google

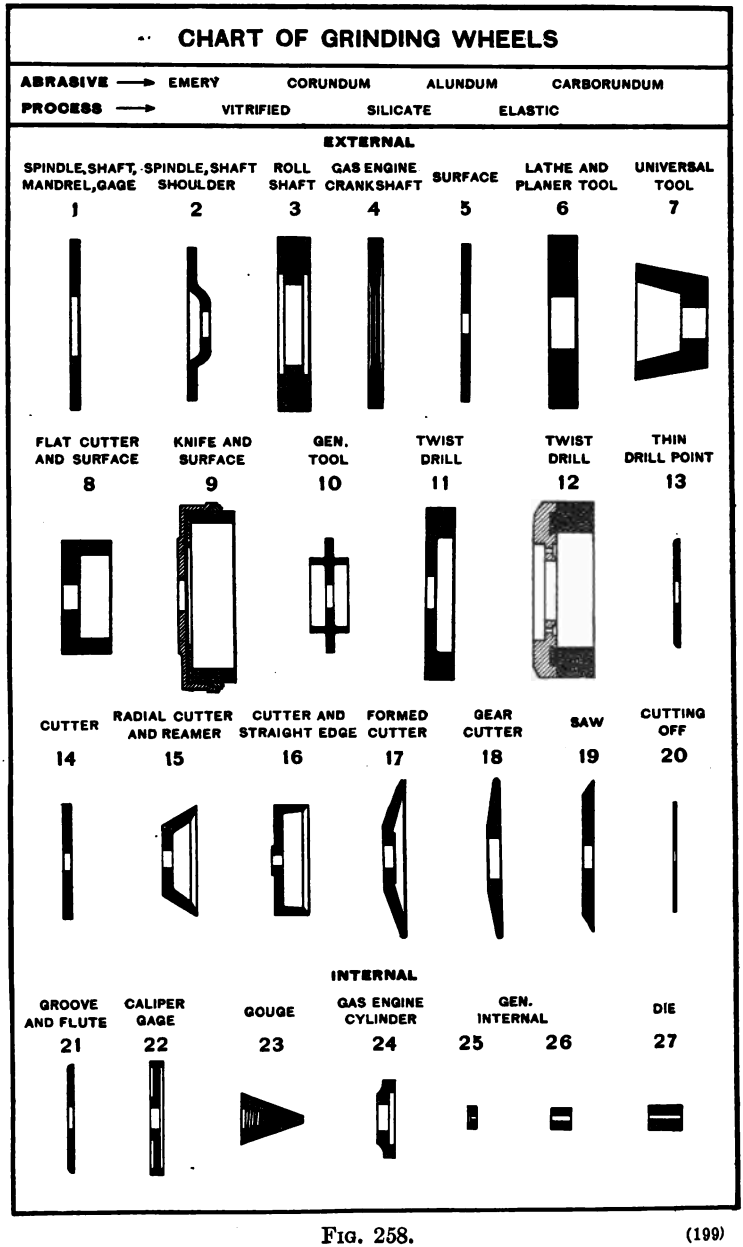


FIG. 258.

Digitized by Google

387. Silicate wheels, light gray in appearance, are made by mixing the abrasive with a bond of silicate of soda, and then subjecting to a low heat (300° to 400° F.). These wheels are made porous or dense, as desired, are even in hardness and have unusual strength. They are adapted for wet tool grinding and general purposes. All wheels above 30″ in diameter are made by this process. See Table of Grades, § 393.

388. Elastic wheels, black in appearance, are made by mixing the abrasive with a bond of shellac, etc.; and then subjecting to a low heat (300° to 400° F.). These wheels are not brittle, and can be made as thin as $\frac{1}{16}$″. They are used for grinding arbors, cutters, reamers for saw gumming, and for cutting off small stock, such as thin steel tubing, wire, etc., also for grinding in narrow openings. See Table of Grades, § 393.

389. Tanite wheels are made by mixing the abrasive with tanite (a chemical and mechanical transformation of leather) and then subjecting to heat under hydraulic pressure. They are used for general grinding in a similar manner to silicate wheels.

390. Vulcanite wheels are made by mixing the abrasive with a bond of rubber and sulphur, and subjecting to a low heat (225° F.). These wheels are not brittle and have a high factor of safety. They are used for special work.

391. Celluloid wheels, brown in appearance, are made by mixing the abrasive with celluloid; both are subjected to a low heat (250° F.).

392. Combination wheels are made by mixing several numbers of the abrasive together in the formation of the wheel. Three numbers are generally used for this purpose, as 24, 36, 46,—the coarsest grade is used for fast cutting and the finest grade is used for a smooth finish. The addition of a small amount of fine abrasive to a coarse wheel will produce wheel of great durability.

Digitized by Google

393. Table of Grades of Wheels Made by Different Processes.

	Vitrified Process.	Silicate Process.	Elastic Process.
Very soft...	E	¾	¾ E
	F	1	1 E
	G	1¼	1¼ E
	H	1½	1½ E
Soft.....	I	1¾	1¾ E
	J	2	2 E
	K	2¼	2¼ E
	L	2½	2½ E
Medium....	M	3	3 E
	N	3½	3½ E
	O	4	4 E
	P	4½	4½ E
Hard.....	Q	5	5 E
	R	5½	5½ E
	S	6	6 E
	T	6½	6½ E
Very hard..	U	7	7 E

Attention. — The grade of a grinding wheel is more important than the number of the abrasive. The number influences the grade to a slight extent, a fine wheel, as No. 90, grade M, is slightly harder than a coarse wheel same grade as No. 60, grade M.

394. Shapes of wheels. — Plain disk wheels are used for straight and taper work, and plain and cup wheels for surface grinding.

Cup, bevel, and disk wheels are used for grinding drills, cutters, and for grinding to shoulder. Small wheels are used for internal grinding. See Chart of Grinding Wheels, Fig. 258.

395. Selection of wheel depends on hardness of material and nature of work. In general, use a hard wheel for rough grinding and a medium or soft wheel for accurate grinding and fine finish. If a wheel is too hard, it will glaze quickly and heat the work; if too soft, it will wear rapidly and prevent accurate grinding. A coarse wheel is less liable to heat than a fine wheel, as the latter is apt to glaze. A perfect wheel is one with a bond that will release the grains before glazing takes place, and hard enough to take a series of cuts without

Digitized by Google

changing its shape or diameter, as accurate grinding depends upon the sizing power of wheel.

396. To mount wheels. — Place rubber or blotting-paper washers between wheels and flanges, and clamp. Wheels should fit easily on arbors. Some are bushed with soft metal to fit standard shafting and will run fairly true. Wheels to be interchangeable without truing are provided with individual taper bushings. Small wheels for internal grinding have counterbore depressions for fillister head screws.

397. The periphery speed for wheels should be approximately from 5000 to 6500 feet per minute (F.P.M.). A higher speed may cause the wheel to break. Should a wheel heat or glaze, run it slower. If it is too soft, it can often be made to hold its size by using a higher speed.

TABLE OF SPEEDS FOR GRINDING WHEELS.

The table given below designates number of revolutions per minute for specified diameters of wheels, to cause them to run at the respective periphery rates of 4000, 5000, and 6000 feet per minute.

DIAM. WHEEL.	REVOLUTIONS PER MINUTE FOR SURFACE SPEED OF 4000 FT.	REVOLUTIONS PER MINUTE FOR SURFACE SPEED OF 5000 FT.	REVOLUTIONS PER MINUTE FOR SURFACE SPEED OF 6000 FT.
1 inch.	15,279	19,099	22,918
2 "	7,639	9,549	11,459
3 "	5,093	6,366	7,639
4 "	3,820	4,775	5,730
5 "	3,056	3,820	4,584
6 "	2,546	3,183	3,820
7 "	2,183	2,728	3,274
8 "	1,910	2,387	2,865
10 "	1,528	1,910	2,292
12 "	1,273	1,592	1,910
14 "	1,091	1,364	1,637
16 "	955	1,194	1,432
18 "	849	1,061	1,273
20 "	764	955	1,146
22 "	694	868	1,042
24 "	637	796	955
30 "	509	637	764
36 "	424	531	637

Digitized by Google

398. The periphery speed of work (F.P.M.) should be proportional to the grade and speed of wheel, also to the diameter of work. The speed varies from 15 to 60 feet per minute for different classes of work. The higher speed is best for cast iron and the lower for duplicate work.

399. Direction of rotation of wheel and work must be opposite at cutting point as in Fig. 259.

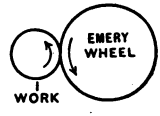


FIG. 259. — DIRECTION OF ROTATION OF WHEEL AND WORK FOR EXTERNAL GRINDING.

400. Feed of table or traverse speed is in proportion to the width of wheel face and finish required. Use coarse feed for roughing. For large work with a heavy machine, use one-half to three-fourths width of wheel to each revolution of work; on light machines use one-third to one-half width of wheel. For a very fine and accurate finish, use one-fourth to one-third width of wheel.

401. Depth of cut. — For roughing take deep cuts, — .001″ to .004″ at each stroke; for finishing light cuts, — .00025″ to .0005″ at each stroke. The sparks thrown off by the grinding wheel indicate the depth of cut: a large volume of sparks indicates a heavy cut, and a small volume a light cut.

402. Width of face of wheel.— A wide wheel with a coarse feed, removes stock rapidly, and is used where it can pass from one-fourth to one-half its width beyond the end of work or recess.

A narrow wheel is used to grind to a shoulder not recessed to produce an accurate diameter next to shoulder or with narrow recess. The face of a wheel may be narrowed by beveling a corner with a diamond tool.

403. To true grinding wheel. — Grinding wheels wear smooth or become glazed by use, so that they will not cut freely. A new wheel will not run true and must be *trued* before using.

Digitized by Google

To true *face* of wheel, mount diamond tool *A*, Fig. 260, in fixture *B*. Feed tool to touch revolving wheel with cross feed, then traverse tool by power or hand long. feed. To true

FIG. 260.— TRUING FACE OF WHEEL WITH A DIAMOND TOOL.

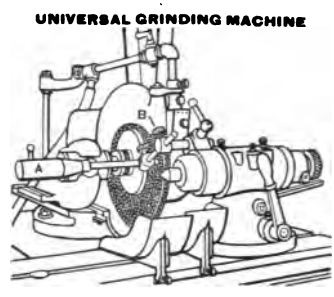


FIG. 261.—TRUING SIDE OF WHEEL WITH A DIAMOND TOOL.

side of wheel, clamp diamond tool *A*, Fig. 261, in slide of fixture *B* at right angles to wheel. Feed tool to wheel by hand long. feed until it touches revolving wheel, then operate tool by hand with cross feed.

If a wheel is soft, but little attention may be given it while roughing, as it will wear away fast enough to keep sharp, but it must be sharp and also true, to produce a fine finish.

Attention. — Grinding wheels may be trued dry, but the wear on the diamond tool is much less if plenty of water is used.

404. To true centers, Fig. 262. — Remove centers, clean holes with waste. Clean dead center and insert it in live center spindle. Belt machine to revolve live spindle, pull out pin *A*, swivel headstock to 30° and clamp. Set reversing dogs and grind center by trial and correction to fit center gage (60°). See § 33. Replace dead center, clean and insert live center in its spindle and grind. Both centers are hardened and tempered. In a plain grinding machine, use a special attachment. Some small grinding machines permit swinging of table to grind centers.

Digitized by Google

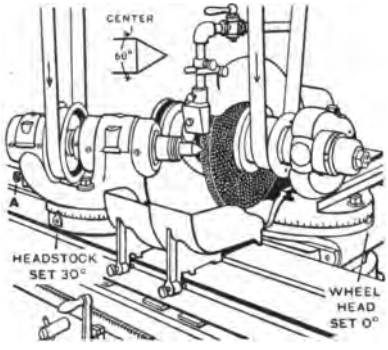


FIG. 262. — GRINDING THE CENTER OF UNIVERSAL GRINDING MACHINE.

405. Methods of driving work. — (Two dead centers.) Centered work is ground on two dead centers and driven by dead center pulley and balanced dog, as in Fig. 256. To

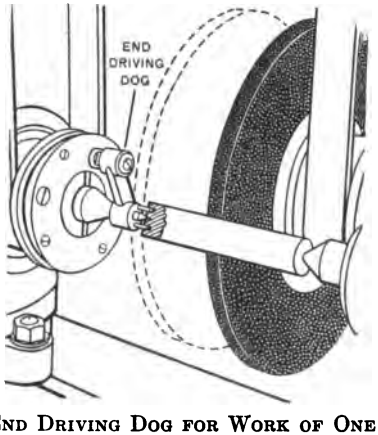


FIG. 263. — END DRIVING DOG FOR WORK OF ONE DIAMETER.

grind work of one diameter, use end driving dog, as in Fig. 263. (With revolving spindle.) For work held in chuck, as face and internal grinding, the spindle revolves.

Some classes of centered work may be rapidly handled and ground by using spindle and triangular live center drive, as in Fig. 264.

Digitized by Google

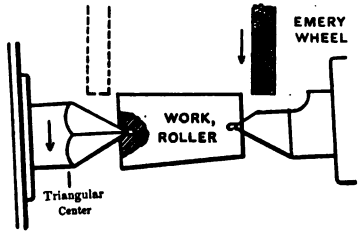


FIG. 264. — TRIANGULAR CENTER DRIVE FOR RAPID PRODUCTION.

A triangular center punch is driven lightly into one center hole to make it fit center. The work is then mounted on centers and not moved until finished.

406. To set swivel table to grind straight work. — Set both zero lines on headstock swivel (graduated in degrees) to coincide, then set both zero lines on swivel table (graduated scale in inches per foot and degrees) to coincide. To obtain accurate setting, take a few light cuts and caliper both ends with micrometer. If they differ, release clamp bolts and swing swivel table a little with adjusting screw at end, clamp bolts and repeat above process until desired accuracy is obtained.

Attention. — The graduations on headstock swivel and swivel table are helpful, but the coincidence of lines, even if located with a magnifying glass, give only approximate accuracy, and it is only by the process of trial and correction that accuracy is obtained.

407. Slight tapers are obtained by setting swivel table to approximate taper by scale and using a standard taper ring gage to determine exact taper, which is obtained by trial and correction in the same manner as a straight setting.

408. Steep tapers on work held in chuck or on headstock spindle are obtained by swiveling the headstock. For work mounted on centers, tapers are obtained by setting wheel slide, and for two abrupt tapers, outside or inside, by setting both headstock and wheel slide at proper angles.

409. Wet and dry grinding. — Wet grinding is rapid and accurate. An insufficient or fluctuating supply of water will cause a change of temperature and produce inaccurate work.

Digitized by Google

If the nature of the work will not permit, or the machine is not arranged for wet grinding, good results may be obtained by grinding dry with very light cuts. Water guards for wet grinding are supplied. See *A*, Fig. 265.

410. Lubricants for grinding. — Use water, and to prevent rusting of machine, add enough sal soda to the water to show a deposit on machine. Machine oil is sometimes added to mixture.

Aluminium is ground with a lubricant of kerosene, or a mixture of kerosene and machine oil.

As most cutter grinders are not arranged for wet grinding, cutters and reamers are ground dry.

411. To prepare work to be ground. Allowance for grinding. — Short work that is not to be hardened is rough turned from .006″ to .010″ large. For large work and long, slender work that will spring and run out of true after turning, and for hardened steel that is liable to spring in hardening, it is good practice to allow from .020″ to .030″ to grind off. A large allowance means but one cut in the lathe, and it is more economical to grind with the modern grinding machine than to take a second cut in the lathe to reduce the allowance.

412. Rough and finish grinding. — In manufacturing machine parts in lots, it is good practice to rough grind all pieces to within .002″ or .003″ before finishing any. The wheel then need be trued only twice, — once in roughing and once for finishing.

413. The finish of work. — An ordinary shaft should be given a "commercial" finish; that is, it should be ground to a good smooth surface. A forcing fit need not be ground to such a fine finish, but close to size. Gages and fine work of this class must be ground both to a fine finish and to an exact size. A fine accurate finish is generally obtained with light cuts, slow feed, a true wheel, and a liberal supply of water.

414. Expansion of work while grinding. — Expansion is caused by the friction of the wheel. In grinding work dry, particularly long shafts, it sometimes is made to run out of

Digitized by Google

true and this is due to its own internal strains or unequal expansion. A grinding operation shows error or truth; an *uneven* volume of sparks indicates error, and an *even* volume of sparks truth of work. To remedy this, flood work with water, reduce speed of work and feed one-fourth thousandth at each end of stroke until work is cylindrically true. When the work becomes round again, increase its speed and the depth of cuts. Spring back rests *B*, *B*, *B*, Fig. 273, are often used to remedy this error.

Attention. — A distinct difference in the sparks, however, may indicate an error as small as one-tenth to one-fourth thousandth of an inch or less, which may be ignored for some classes of work.

415. Seasoning work. — Long slender work is often laid away for a period of time between roughing and finishing, to relieve internal strains, when great accuracy is required. This is usually unnecessary for short or heavy work.

416. Care of machine and work. — All bearings should be kept well oiled. The cross slide should be cleaned, oiled, and if need be, adjusted to move smoothly. The centers and center holes in work must be kept absolutely true and clean and well oiled to produce accurate work. While most of the bearings on modern grinding machines are dust-proof, nevertheless the machines should be kept clean.

417. Measuring tools for grinding — Use micrometer calipers and limit gages for general work. For very accurate work use a ten-thousandth micrometer or measuring machine.

Warning. — Push the shipper slowly and watch the grinding wheel as it slowly starts. Never stand in front of a wheel when it is starting, to avoid injury if the wheel should happen to break. Modern grinding wheels have a high factor of safety and seldom burst at the speed recommended, but any wheel may be broken by careless usage; accidentally moving the headstock, or footstock, or work, against the side of the wheel, especially a thin, vitrified wheel, revolving or stationary, may break it.

Digitized by Google

PROBLEMS IN CYLINDRICAL GRINDING.

418. Adjustments and movements to operate universal grinder, Fig. 265.

SCHEDULE OF PARTS.

1. Shipper to start and stop grinding wheel.
2. Shipper brake to start and stop work.
3. Knob to start or stop long. power feed work (table feed).
4. Long. feed hand wheel.
5. Lever to reverse long. feed by hand.
6 and 7. Dogs to reverse long. feed automatically.
8. Cross feed hand wheel.

Automatic Cross Feed.

9. Pawl.
10. Ratchet wheel (each tooth reduces work 1/4/1000″ in diameter and each graduation 1/1000″).
11 and 12. Adjusting screws which control movements of pawl 9 and depth of cut at each end of stroke.
13. Perpendicular latch.
14. Latch head.
15. Shield.
16. Horizontal latch.

Swivel Table.

17 and 18. Bolts to clamp swivel table.
19. Screw to adjust swivel table. To grind straight or taper, see Graduated Scale at end of swivel table.

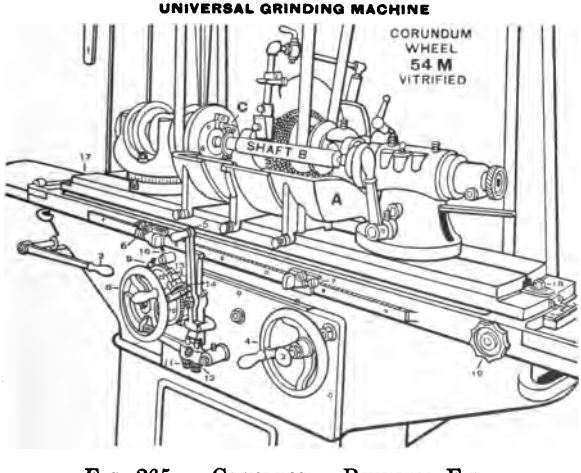


FIG. 265. — GRINDING A RUNNING FIT.

Digitized by Google

419. To grind $1\frac{1}{16}''$ running fit (wet), universal grinding machine, Fig. 266.

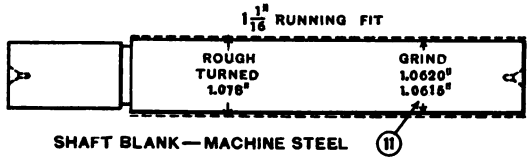


FIG. 266. — SCHEDULE DRAWING OF GRINDING A RUNNING FIT.

SCHEDULE OF OPERATIONS.

ROUGHING. FINISHING.

Shaft blank, machine steel, rough turned 1.0780″ diameter.

Corundum grinding wheel 12″ × ¾″, No. 54, Grade M, vitrified.

Speed of wheel 5000 F.P.M. Speed of work 50 F.P.M. Feed ⅛ width wheel per revolution.

Oil bearings of machine with machine oil.

True wheel (see § 403) and centers (see § 404). Set machine to grind straight. See § 406.

1. Grind to diameter, (**11**). Arrange water guards *A*, Fig. 265, for grinding.

2. Unlock horizontal latch 16, from latch head 14 throw out pawl 9 and move grinding wheel back with cross feed wheel 8, then mount shaft *B* on centers. Use special grinding dog *C* with copper under set screw. Start wheel, work and feed 1, 2, 3, and adjust table dogs 6 and 7 to obtain length of stroke and avoid wheel striking dog or footstock.

3. Set automatic cross feed at 11 and 12 to feed $\frac{1}{1000}''$ at each end of stroke (4 teeth on ratchet). Move grinding wheel to lightly cut revolving work with hand cross feed wheel 8. Throw pawl 9 to mesh with 10, and take about two trial cuts entire length of work.

4. Throw out pawl 9 and allow wheel to pass over work several times until cutting nearly dies out without moving cross feed, then stop work with grinding wheel at footstock end with shipper brake 2, and measure work with 2″ micrometer.

Subtract rough diameter of work 1.0620″ + .002″ for finishing from reading of micrometer. For example, reading of micrometer may be 1.0740″; then 1.0740″ − 1.0640″ = .0100″ for rough grinding. As each tooth in ratchet wheel 12 = 1/4/1000″ in diameter of work and work is .010″ large, the work is 40 teeth large.

5. Rough grind .002″ large as follows: Raise perpendicular latch 13 in head 14, and without moving cross feed, move shield 15 to right

Digitized by Google

SCHEDULE OF OPERATIONS. *Concluded.*

until end of shield and pawl tooth are 40 teeth apart, then drop in latch 13. Start work (2) when shield 15 throws out pawl 9, stop work and measure, and from reading subtract 1.0620″. (1.0640″ − 1.0620″ = .002″.)

6. Finish grind with pinch feed 1/4/1000″ in diameter of work as follows: Lock latch head 14 with latch 16, start work. Take two cuts 1/2/1000″ each (two pinches at each end of work). Stop work and measure (1.0630″). Take one cut 1/4/1000″ (one pinch at footstock end of work) and measure and repeat until work measures 1.0620″.

Limit 1.0620″ to 1.0615″.

This shaft is for a running fit in $1\frac{1}{16}$″ hole. Hole limit 1.0625″ + or − .00025″. See Running Fits, § 137. Clean machine with waste.

Attention. — After the student has ground a number of problems and become familiar with the behavior of machine and the automatic feed, he may let wheel pass over work after trial cuts until cutting sparks die entirely out, then set automatic feed to finish grind to diameter without using pinch feed except for wear of wheel.

Note. — The student should examine wheel and centers and, if in good condition, truing may be omitted.

420. Adjustments and movements to operate universal and tool grinding machine, Fig. 267.

SCHEDULE OF PARTS.

1. Shipper to start and stop grinding wheel.

2. Shipper brake to start and stop work.

3. Lever to start or stop long. power feed (table feed).

4. Long. feed hand wheel.

5. Lever to reverse long. feed by hand.

6 and 7. Dogs to reverse long. feed automatically.

8. Cross feed hand wheel. Each graduation reduces work 1/1000″ in diameter.

Fine Cross Feed.

9. Thumb nut for controlling fine or coarse cross feed.

10. Thumb screw for operating fine cross feed.

Swivel Table.

17. Bolt for clamping swivel table.

18. Spring knob for quick adjustment of swivel table.

19. Thumb screw for fine adjustment of swivel table. To grind straight or taper, see Graduated Scale at end.

Grinding Wheels.

20. Hand wheel to elevate grinding wheel.

Digitized by Google

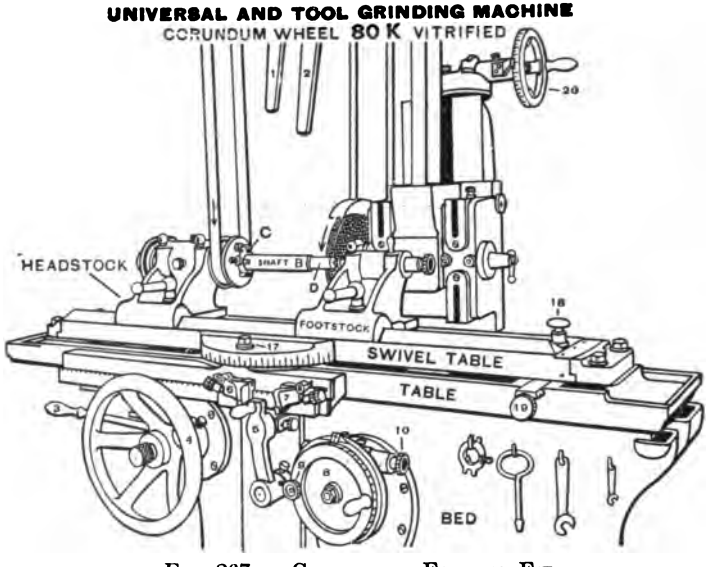


FIG. 267. — GRINDING A FORCING FIT.

421. To grind 1″ forcing fit (dry) with a universal and tool grinding machine, Fig. 267.

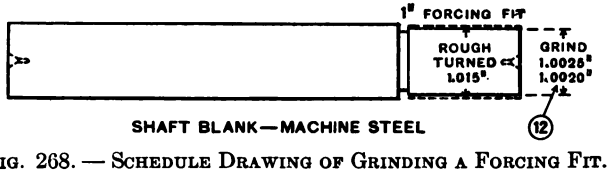


FIG. 268. — SCHEDULE DRAWING OF GRINDING A FORCING FIT.

SCHEDULE OF OPERATIONS.

ROUGHING. FINISHING.

Shaft blank, machine steel, rough turned, 1.075″ diameter.

Corundum grinding wheel 7″ × ½″, No. 80, Grade K, vitrified.

Speed of wheel 5000 F.P.M. Speed of work 50 F.P.M. Feed ⅓ width wheel per revolution.

Oil bearings of machine with machine oil.

True wheel and centers. Set machine to grind straight.

1. Grind to diameter, (**12**), Fig. 268. Loosen nut 9, Fig. 267, and move grinding wheel back with cross feed wheel 8, then mount shaft *B* on centers. Use grinding dog *C* with copper under

Digitized by Google

SCHEDULE OF OPERATIONS. *Concluded.*

set screw. Start grinding wheel, work and feed 1, 2, 3, and adjust dogs 6 and 7 to obtain length of stroke of portion *D* and avoid wheel touching shoulder of work.

2. Move grinding wheel to lightly cut revolving work with coarse hand cross feed wheel 8. Tighten nut 9 and use fine cross feed thumb screw 10. Graduations on cross feed reduce work 1/1000″ in diameter. Take two or three trial cuts 1/2/1000″ each, feeding at footstock end only, and allow wheel to pass over work until cutting nearly dies out, then stop table and work with grinding wheel at footstock end with shippers 3, 2, and measure work with a 1″ micrometer.

3. Rough grind .002″ large as follows: Use 1/2/1000″ cuts with thumb screw 10 at footstock end only, and grind to 1.0025″ + .002″ = 1.0045″.

For example, reading of micrometer may be 1.012″, then take 8 cuts and measure (1.008″); take 4 cuts and measure (1.0060″); take 3 cuts and measure (1.0045″).

4. Finish grind as follows: Use 1/4/1000″ cuts and feed at footstock end only, as follows: Take 2 cuts and measure (1.0040″); take 3 cuts and measure (1.00325″); take one cut and measure, and repeat until work measures 1.0025″.

Limit 1.0025″ to 1.0020″.

Attention. — It is to be forced into a 1″hole. Hole li mit 1.000″ + or − .00025″. See Forcing Fits, § 140.

Clean machine with brush and waste.

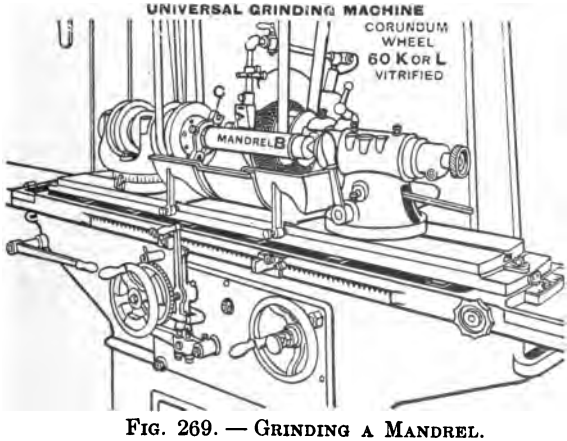


FIG. 269. — GRINDING A MANDREL.

422. To grind a $\frac{7}{16}$″ standard mandrel, taper .006″ to 1,″ Figs. 269 and 270.

Digitized by Google

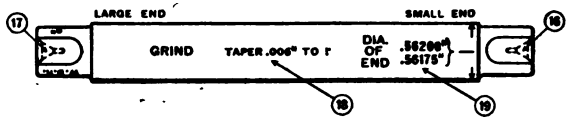


FIG. 270. — SCHEDULE DRAWING OF GRINDING $\frac{9}{16}$″ STANDARD MANDREL.

SCHEDULE OF OPERATIONS.

ROUGHING. FINISHING.

Mandrel blank, carbon steel, smooth turned .580″ diameter.

Stamped, hardened, and ends tempered. See *Elements of Machine Work.*

Corundum grinding wheel 12″ × ½″, No. 60, Grade K or L, vitrified.

Speed of wheel 5000 F.P.M. Speed of work 50 F.P.M. Feed ⅓ width of wheel per revolution.

Oil bearings of machine with machine oil.

True wheel and centers. Set swivel table to grind the taper.

---

1. Lap center holes, (**16**), (**17**). See *Attention* § 310. Grind to taper, (**18**). Grind to diameter, (**19**). Arrange water guards. See Fig. 265.

2. Move grinding wheel back with hand cross feed wheel 8.

Mount mandrel *B* on centers with dog *C* on stamped end without copper under set screw. Start grinding wheel, work and feed (1, 2, 3), and adjust dogs 6 and 7 to length of stroke.

3. Set automatic feed at 11 and 12 to feed 1/1000″, four teeth at each end of work. Move grinding wheel to lightly cut revolving work with hand cross-feed wheel 8, then throw pawl 9 in mesh and take three or four cuts with automatic feed. Then throw out pawl 9 and allow wheel to pass over work until cutting nearly dies out. Then stop work with grinding wheel at footstock end and measure both ends with micrometer. To correct taper, unclamp nuts 17 and 18 and move swivel table by screw 19: take one or two cuts with automatic cross feed. Measure and repeat until taper is correct.

4. Rough grind .002″ large, as follows: Measure small end, then subtract roughing diameter .562″ + .002″ for finishing from reading of micrometer. For example, reading of micrometer may be .574″ − .564″ = .010″ for rough grinding. Set shield 15 and pawl 9, 40 teeth apart and drop in latch 13. Start work (shipper 2), and when machine stops feeding, stop work and measure, and from reading subtract .562″ (.564″ − .562″ = .002″).

5. Finish grind with pinch feed as follows: Lock latch head 14 with horizontal latch 16.

Digitized by Google

SCHEDULE OF OPERATIONS. *Concluded.*

Start work and pinch feed to finish. Take two cuts 1/2/1000 each (two pinches at each end of work), stop work and measure. Take one cut 1/4/1000″ (one pinch) at footstock end of work and measure and repeat until work measures .562″. Limit .56200″ to .56175″. Clean machine with waste.	

423. To grind a shoulder, Fig. 271.

Spindle blank, crucible cast steel.

Alundum grinding wheel (plain) 12″ × ½″, No. 60, Grade K, vitrified.

Speed of wheel 5000 F.P.M. Speed of work 50 F.P.M.

Feed, ⅓ width of wheel.

True wheel and centers. Set machine to grind straight.

Mount spindle blank *A*, recessed as at *B* for side clearance, on centers. Control depth of cut by long. feed, and feed by

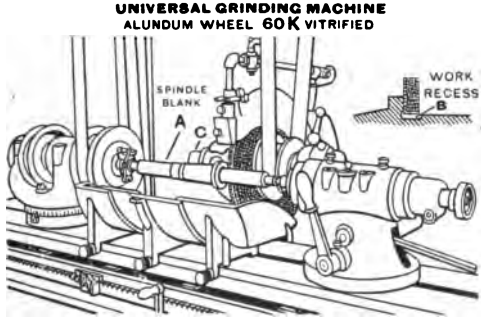


FIG. 271. — GRINDING A SHOULDER.

cross feed. If shoulder is large use special wheels mounted on end of wheel spindle *C* and set wheel swivel ½° from 90° to give clearance.

424. Plain grinding machine for commercial grinding, Fig. 272. — The essential parts of this machine are similar to those of a universal grinding machine except that the headstock and wheel slide do not swivel, and the swivel table forms a water guard. It is designed for straight and taper outside work only. It is made strong and rigid to grind work rapidly.

Digitized by Google

Some plain grinding machines have a traveling grinding wheel instead of a traveling table.

425. To grind cast-iron roll, *A*, Fig. 272.

Roll blank, cast iron. Rough turned .010″ to .020″ large.

Alundum grinding wheel (plain) 20″ × 2″, No. 24 Combination, Grade K, vitrified.

Speed of wheel 5000 F.P.M. Speed of work 50 F.P.M.

Feed, ¾ width of wheel.

True wheel and centers. Set machine to grind straight.

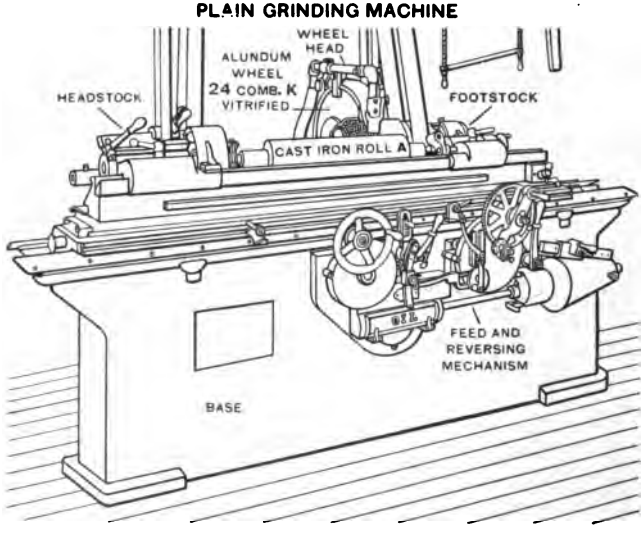


FIG. 272. — GRINDING A CAST-IRON ROLL.

Set automatic feed to feed at both ends of stroke and to grind .002″ large, then use pinch feed to finish.

Mount roll *A* on dead centers.

Place universal back rests 6 to 10 diameters apart and grind to required diameter with water.

426. To grind slender shaft, *A*, Fig. 273.

Shaft blank, machine steel smooth turned .012″ large.

Alundum grinding wheel (plain) 12″ × ¾″, No. 60, Grade L vitrified.

Digitized by Google

Speed of wheel 7000 F.P.M. Speed of work 90 F.P.M.

Feed, $\frac{3}{4}$ width of wheel.

True wheel and centers. Set machine to grind straight.

Set automatic feed to feed at both ends of stroke and to grind .002″ large, then use pinch feed to finish.

Mount shaft *A* on dead centers. Place three universal back rests *B* against shaft, 6 to 10 diameters apart, see Fig. 273, and grind to required diameter.

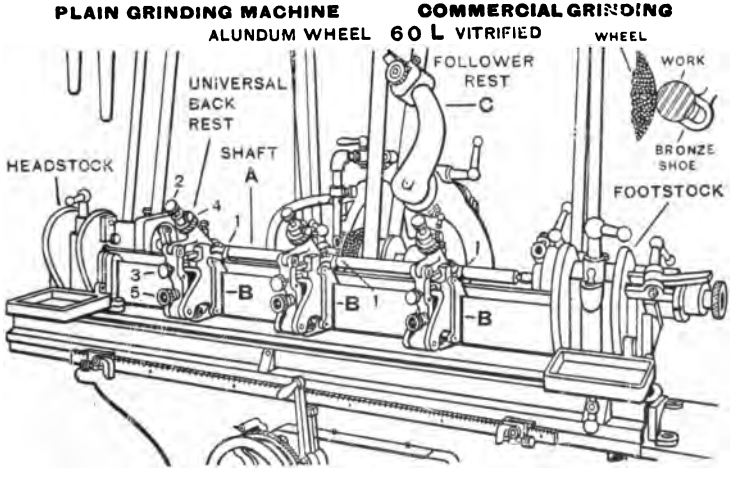


FIG. 273. — GRINDING A SLENDER SHAFT.

427. Back rests, plain and universal, are fixtures used to reduce vibration and permit a greater depth of cut on straight and taper work both large and small.

The plain rest has a single shoe of wood or soft metal and is used for small, short work.

The universal rests *B*, *B*, *B* can be delicately adjusted. Select bronze shoes 1, 1, 1, (see detail) the size of finished work, move screws 2 to maintain contact of shoe upon work while grinding trial piece, and adjust stop screw 3 to preserve diameter when finished.

Adjust screws 4 and 5 to regulate pressure of springs upon shoes and work.

Digitized by Google

Duplicate pieces are ground without disturbing adjustments except slightly for wear of shoe 2 and diameter 3.

Follower rest *C* is used on slender work that has been rough ground .001″ large. It cannot be used on taper work. Adjust with wheel and work in motion.

428. Automatic magnetic sizing grinder, Fig. 274. —

The mechanical automatic feeding mechanism is electrically controlled and duplicates straight and taper work regardless of wear of grinding wheel.

A coarse feed for roughing automatically switches to a fine feed for finishing, and when work is finished to size the feed automatically stops.

429. To rough and finish grind spindles in an automatic magnetic sizing grinder, Fig. 274. **Duplicate work.** —

Spindle blank, machine steel, rough turned 0.020″ large.

Corundum grinding wheel (plain) 18″ × 1½″, No. 40, Grade M, vitrified.

Speed of wheel 5000 F.P.M. Speed of work 50 F.P.M.

Feed, ¾ width of wheel.

True wheel and centers. Set machine to grind straight.

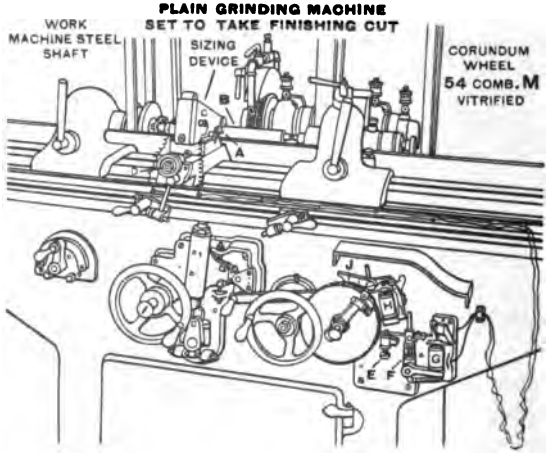


FIG. 274. — AUTOMATIC SIZING DEVICE FOR GRINDING DUPLICATE WORK.

Digitized by Google

To set automatic sizing device, rough and finish grind one piece to required size with automatic mechanical feed, and use this as a master piece to set sizing device. Place diamond-point bearer *A*, on master piece *B* and adjust arm *C* by screw *D* to make an electrical contact. Then mount blank for next piece on centers. Adjust coarse feed for roughing by screw *E* to feed .001″ at each end of stroke, and adjust screw *F* to have magnet *G* throw out the coarse and throw in the fine feed one tooth (one-eighth thousandth) when piece is within .002″ of size.

Magnet *H* throws out latch *J* and stops feed when piece is gound to correct diameter.

Attention. — The coarse and fine feeds and the amount allowed for finishing may be varied for different classes of work. Dry batteries or a plug in a lamp socket of a direct-current circuit will operate device.

430. To grind a straight and taper bearing on a spindle, Fig. 275. —

Spindle blank, crucible steel unannealed. Rough turned .010″ to .020″ large.

Corundum grinding wheel (plain) 12″ × 1″, No. 54, Grade M, vitrified.

Speed of wheel 5000 F.P.M. Speed of work 50 F.P.M.

Feed, $\frac{1}{3}$ width of wheel.

True wheel and centers.

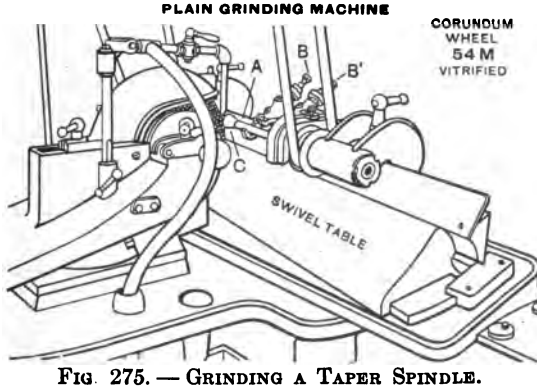


FIG. 275. — GRINDING A TAPER SPINDLE.

Digitized by Google

Set automatic feed to feed .001″ at both ends of stroke and to grind .002″ large, then use pinch feed to finish.

Mount spindle *A* on dead centers.

Place two universal back rests *B* and *B′* against straight portions of work. After straight portions are ground, set swivel table to grind taper bearing *C*. To obtain correct taper, take light cuts, try taper in box and adjust table until taper fits. Use "artists'" blue for marking in taper hole.

431. To grind taper collet, *A*, Fig. 276.—

Collet blank, machine steel. Rough turned .010″ to .020″ large and recess at *B*.

Corundum grinding wheel (plain) 12″ × ½″, No. 54 Combination, Grade M, vitrified.

Speed of wheel 5000 F.P.M. Speed of work 50 F.P.M.,

Feed, ¾ width of wheel.

True wheel and centers.

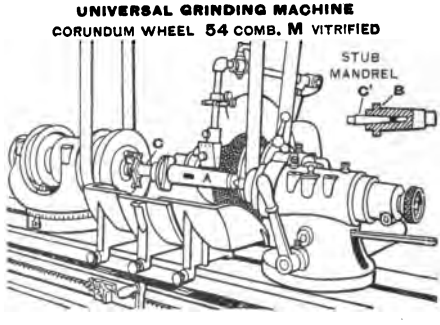


FIG. 276. — GRINDING A TAPER COLLET.

Set automatic feed to feed at both ends of stroke and to grind to .002″ large, then use pinch feed to finish.

Place stub mandrel *C*, *C′* in taper hole of collet *A*, mount on centers.

Set swivel table to grind taper. To obtain correct taper, take light cuts, try taper in gage or spindle hole and adjust table until taper fits. Use "artists'" blue for marking in hole.

Digitized by Google

432. To grind a phosphor bronze taper bushing, *A*, Fig. 277. —

Bushing blank, phosphor bronze. Rough turned to enter taper hole within $\frac{3}{8}''$ of small end.

Corundum grinding wheel (plain), $12'' \times \frac{1}{2}''$, No. 54, Grade M, vitrified.

Speed of wheel 7000 F.P.M. Speed of work 75 F.P.M.

Feed, $\frac{1}{3}$ width of wheel.

True wheel and centers.

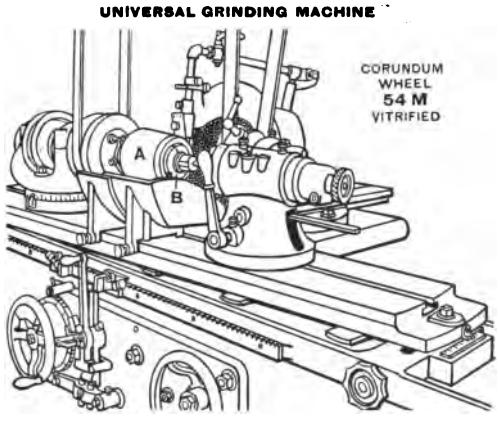


FIG. 277. — GRINDING A BRONZE TAPER BUSHING.

Set automatic feed to feed .001″ at both ends of stroke and to grind .002″ large, then use pinch feed to finish.

Place bushing *A* on a built-up taper mandrel *B* and mount on dead centers.

Set swivel table to grind taper. To obtain correct taper, take light cuts and test taper in taper hole in frame, and adjust table until taper fits.

Digitized by Google

CHAPTER XV.

DRILLING MACHINES. VERTICAL DRILLING MACHINE. TWIST DRILLS. GRINDING DRILLS. SIZES OF DRILLS. SPEEDS AND FEEDS OF DRILLS. LAYING OUT AND DRILLING HOLES. DEEP DRILLING.

DRILLING MACHINES.

433. Drilling machines are divided into two general classes, vertical and horizontal. See *Advanced Machine Work.*

Among the first are centering, vertical, multiple spindle, and plain radial and universal radial drills. Among the second, centering, multiple spindle, and horizontal drilling and boring machines.

There are many special machines in both classes, such as portable drills, drills driven by hand, rope, and belt, flexible shaft drills, pneumatic drills, and electrically-driven drills.

Speeds. — Drilling machines with belt and cone drives usually give eight speeds — four direct and four with back gears. Machines with speed gear boxes usually have sixteen speeds — eight direct and eight with back gears. See § 458.

Feeds. — Drilling machines with belt and cone feeds usually give from three to four feeds; and with gear feed boxes, from three to six feeds.

VERTICAL DRILLING MACHINE.

434. Vertical drilling machine described, Fig. 278. — This machine, commonly called a drill press, is an indispensable part of a modern manufacturing equipment, and is used for all ordinary drilling operations. On this type of machine the work remains stationary while the revolving drill is fed downward through the work by hand or by power.

Digitized by Google

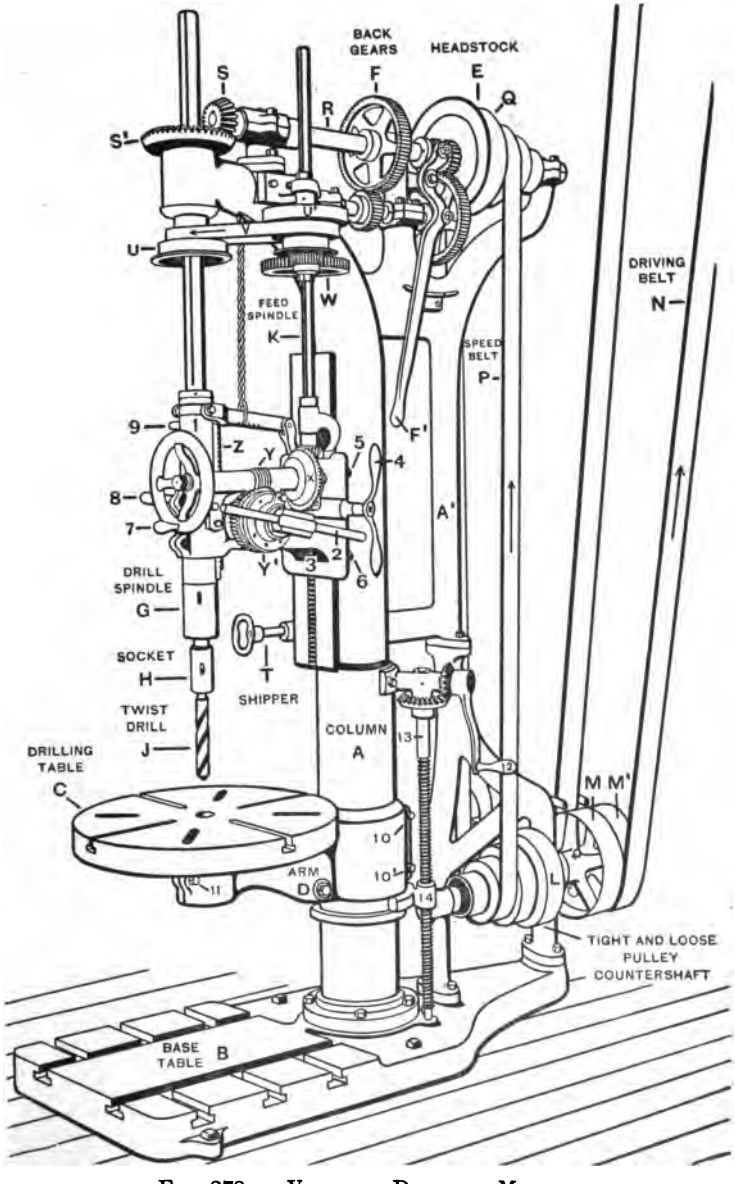


FIG. 278. — VERTICAL DRILLING MACHINE.

Digitized by Google

SCHEDULE OF PARTS.

TIGHT AND LOOSE PULLEY DRIVE. CONE HEADSTOCK. BELT FEED.

A and *A'* — Double column.

B — Base table for large work, slotted to hold clamping bolts.

C — Drilling table, revolving.

D — Arm supporting table swiveled on column *A*.

E — Headstock.

F and *F'* — Back gears and lever to throw back gears "in" or "out."

G — Drill spindle.

H — Socket for drill.

J — Drill.

K — Feed spindle, splined.

L — Cone pulley, four steps.

M and *M'* — Tight and loose pulleys.

N — Driving belt.

P — Speed belt.

Q — Headstock cone pulley.

R — Spindle.

S and *S'* — Bevel gear fast to spindle and bevel gear feather keyed to drill spindle. The drive is from *N* to *M* through *L*, *P*, *Q*, *R*, *S*, *S'* to spindle or with back gears in *N*, *M*, *L*, *P*, *Q*, through back gears to *F*, *R*, *S*, *S'* to spindle.

T — Shipper rod.

U and *U'* — Feed cones, three steps.

V — Feed belt.

W — Spur gears.

X — Bevel gears.

Y and *Y'* — Worm and worm gear.

Z — Rack. A pinion on spindle of worm gear *Y'* meshes with rack.

1. Sleeve, in which spindle *G* rotates; its weight and that of the spindle is counterbalanced by weight in column attached to chain shown. It also carries rack *Z*. The feed motion is through *U*, *V*, *U'*, *W*, *K*, *X*, *Y*, *Y'* to *Z*.

2. Lever to move spindle quickly up or down in head to lock or unlock clutch throwing in or out hand and power feed.

3. Head, position adjustable up or down column.

4. Lever to adjust head.

5. Clamp bolt.

6. Clamp bolt. By unclamping 5 and 6 the head 3 can be adjusted by lever 4.

7. Hand feed wheel. For hand feed, lock friction clutch by pressing lever 2 inward, then turn wheel.

8. Power feed lever. For power feed, bring drill to work and lock clutch by lever 2, then lock clutch in gear *X* by pressing lever 8 upwards.

9. Stop, position adjustable.

10 and 10'. Clamp bolts. By unclamping bolts the table *C* may be swung around column *A*.

11. Clamp bolt; by unclamping may revolve table about its center.

12. Elevation handle.

13. Elevation screw.

14. Nut threaded to 13 and fitted to arm.

To adjust table vertically, loosen 10 and 10' with handle 12, turn screw 13 in nut 14.

Digitized by Google

TWIST DRILLS.

435. Twist drills are made by milling two helical grooves of increase twist in a carbon or high-speed steel-drill blank (a round bar) to form cutting edges, as in Fig. 279, and to carry away the chips. For Three and Four-Groove Drills, see § 478. For Drills Twisted from a Flat Bar, see § 444.

The metal between the grooves is the web, and is thicker at shank than at point to make the drill stiffer. The cutting edges are *A*, *B*, and *C*, Fig. 279. A drill has three clearances. Body and land clearances are given the drill in manufacturing. The body clearance is obtained by turning or grinding the body slightly smaller at the shank end; and the land clearance by milling, leaving narrow lands as at *DD'* and *EE'*.

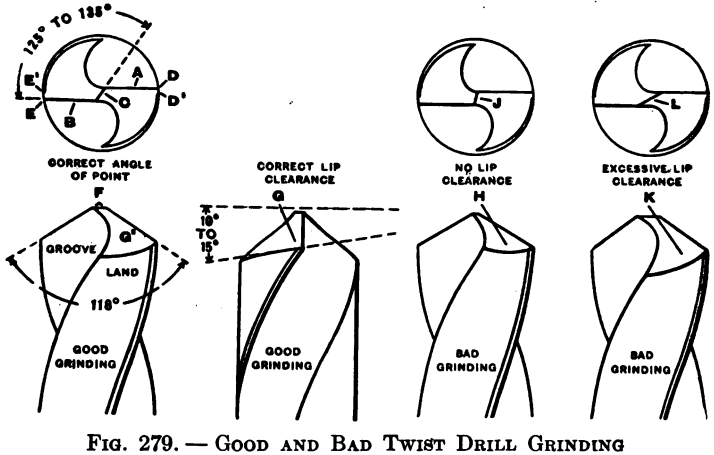


FIG. 279. — GOOD AND BAD TWIST DRILL GRINDING

When a drill is ground to an angle of 118°, as at *F*, the cutting edges *A* and *B* are straight and correct, but if ground to any other angle they will be curved and likely to produce an irregular hole. Some drill manufacturers use an angle of 126° for the point of the drill instead of 118°.

The lip clearance, *GG'*, is obtained by the student in grinding the drill with a twist drill grinder, Fig. 282, or by hand, Fig. 283.

Digitized by Google

The angle of point C, Fig. 279, is an indication of the clearances and should form an angle from 125° to 135° with lip B. This gives a lip clearance at G and G' from 10° to 15°, gradually increasing from periphery to center. Drill H is badly ground. Having no clearance, as indicated by angle of point J, it will not cut and may split at the web. Drill K is also poorly ground. Having too much clearance, as indicated by angle of point L, it will cut but will dull quickly and may chatter.

For very hard material, as unannealed carbon steel, a drill will cut more effectively if given a small lip clearance, and for soft material, as gray iron castings, it will cut more rapidly if given a larger clearance.

Attention. — A reamer drill, that is, a drill that is to be followed by a reamer, should be carefully ground, and it is safest to test it in a trial piece, particularly if the drill is ground by hand, to see if it allows sufficient for reaming.

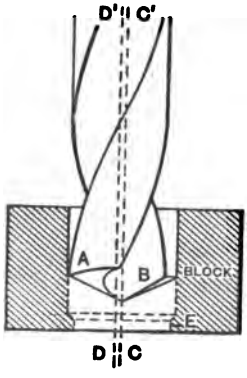


FIG. 280. — DRILL CUTTING, LARGE, BADLY GROUND.

436. The effect of a long and short lip, Fig. 280. — A drill ground with lip A longer than lip B will cut large, the drill following in the path of its point CC', not in the geometric center line DD'. When point pricks through, the drill rights itself and drills the remainder of the hole to the proper size as at E.

FIG. 281. — TWIST DRILL WITH POINT THINNED BY GRINDING.

437. Thinning the point of a twist drill. — As a drill is sharpened by grinding, the point becomes thicker on account of the increasing thickness of the web and requires greater pressure to make it cut. To remedy this, grind a groove C, Fig. 281, on each side with the thin wheel of the grinder C, Fig. 282.

Digitized by Google

GRINDING DRILLS

438. Machine method of grinding a twist drill, Fig. 282.

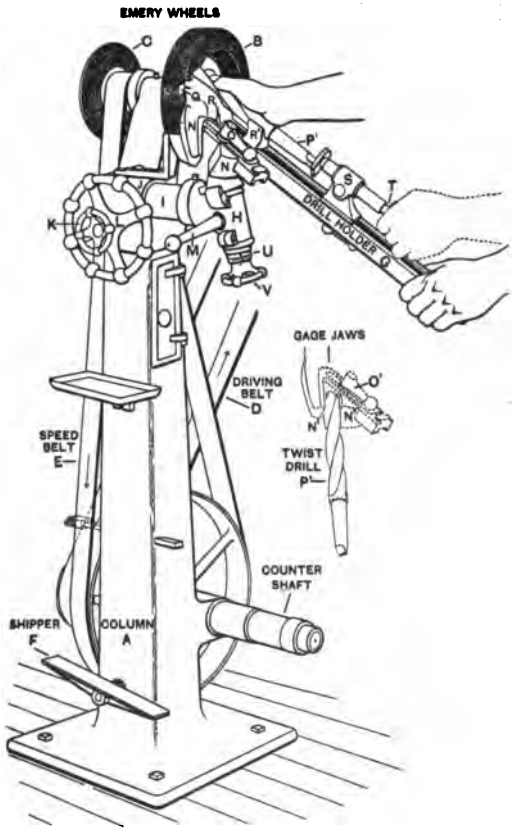


FIG. 282. — GRINDING TWIST DRILL WITH TWIST-DRILL GRINDER.

SCHEDULE OF PARTS AND OPERATIONS.

A — Column carrying wheels.	*E* — Speed belt.
B — Cup-shaped grinding wheel for grinding lips of drill.	*F* — Shipper, operated by the foot.
C — Thin grinding wheel for thinning point of drills. See §437.	*G* — Drill holder, swivels in its bearing.
D — Driving belt.	*H* — Bearing for drill holder.

Digitized by Google

SCHEDULE OF PARTS AND OPERATIONS. *Concluded.*

I — Arm, adjustable.
J — Casing for arm.
K — Clamp wheel.
L — Wheel which carries pinion that engages a rack on under side of arm *I*. To move arm *I* and drill holder in or out, release wheel *K* and revolve wheel *L*.
M — Handle to swing arm sideways.
N — Gage jaws.

To Grind Drill.

O — Handle to unclamp jaws to set for desired clearance as at *N'*, *N'*, *O'*, *P'*, to allow drill to just pass through.
P — Twist drill.
Q — Lip rest; set drill holder forward so that rest will just clear grinding wheel, then swivel in bearing *H* and clamp by wheel *K*.
R and *R'* — V holder. Swing holder to left and place drill with lip against lip rest *Q* and projecting slightly beyond it. Hold in position by hand.
S—Footstock. Move it against end of drill and clamp.
T — Feed screw, to regulate cuts, which should be light. Swing holder to right and cut. After one lip is ground, turn the drill over and grind the second lip, thus making it a counterpart.
U — Eccentric sleeve graduated to obtain more or less clearance for cutting different metals.
V — Wheel to clamp sleeve *U*.

439. To grind a twist drill by hand, Fig. 283. — Place drill *A* on rest *B* against grinding wheel *C*, with its cutting edge in a horizontal position. Lower the left hand slightly to

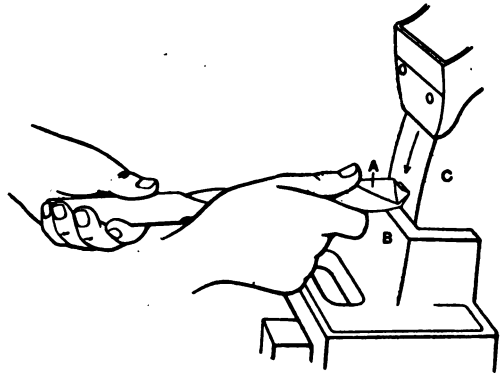


FIG. 283. — GRINDING TWIST DRILL BY HAND.

Digitized by Google

obtain the lip clearance, then turn drill over and repeat on second lip.

Twist drills are obtainable with grinding lines AA' and BB', Fig. 284, which allow for proper angle and location of point. As the lips should be of equal length and height, their length may be measured with a rule as in Fig. 284, and their height as in Fig. 285. Any irregularity in the cutting lips is doubled in action.

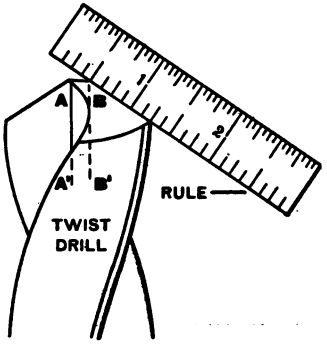


FIG. 284.— MEASURING LENGTH OF THE DRILL LIP.

A more rapid and accurate method of testing the angle of point and length and clearances of lips is to use a twist drill grinding gage, as in Fig. 286. Drill A is placed in V groove B and gage C adjusted central with drill point by screw D.

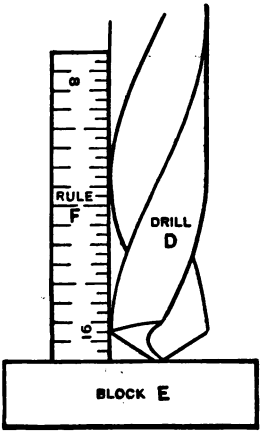


FIG. 285.— TESTING CLEARANCE AND HEIGHT OF CUTTING LIPS OF TWIST DRILL.

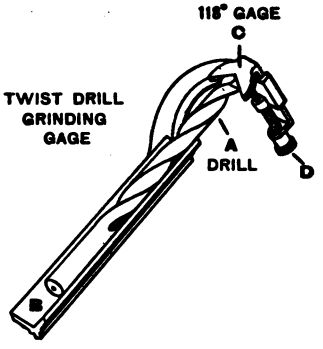


FIG. 286. — TESTING ANGLE OF DRILL POINT.

Attention. — It requires intelligent practice to grind a twist drill correctly by hand.

Digitized by Google

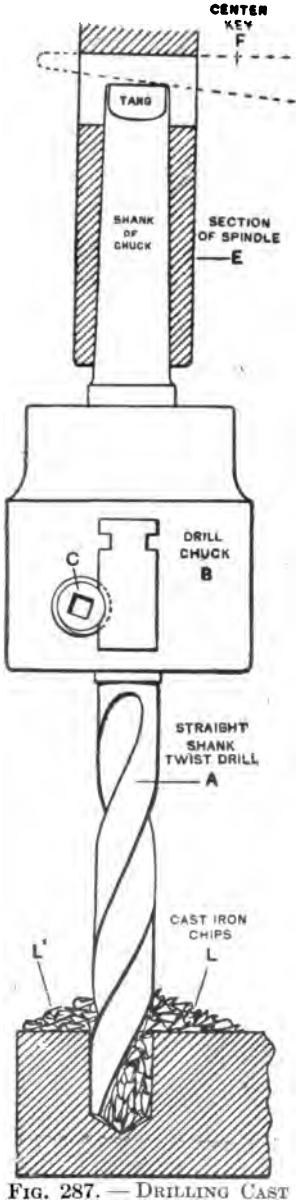


FIG. 287. — DRILLING CAST IRON.

440. To use a straight-shank twist drill, Fig. 287.— Clean shank of chuck and insert in spindle. Place drill *A* in chuck *B* and clamp by screw *C* operated with wrench *D*, Fig. 288. To remove chuck from spindle *E*, drive in key *F* lightly.

441. Drilling cast iron with a twist drill. — The cast-iron chips *L* and *L'* will come out of the hole in large and small fragments if the drill is properly ground otherwise the chips will be very small particles or reduced to powder.

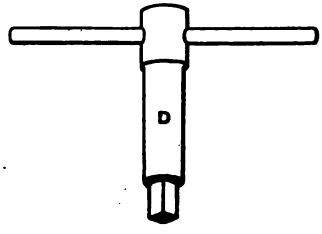


FIG. 288. — DRILL CHUCK WRENCH.

Attention.— A drill will bore a hole slightly larger in cast iron than in steel, wrought iron, brass or copper.

442. To use a taper shank drill, Fig. 289. **See Morse Standard Tapers,** §§ 648, 649. — Clean shank of socket *H* and insert in spindle *I*, and clean shank of drill *G* and insert in socket. To remove drill *G*, drive center key in slot lightly.

443. Drilling steel or wrought iron with a twist drill, Fig. 289.— Steel and wrought iron chips *M* and *M'*

Digitized by Google

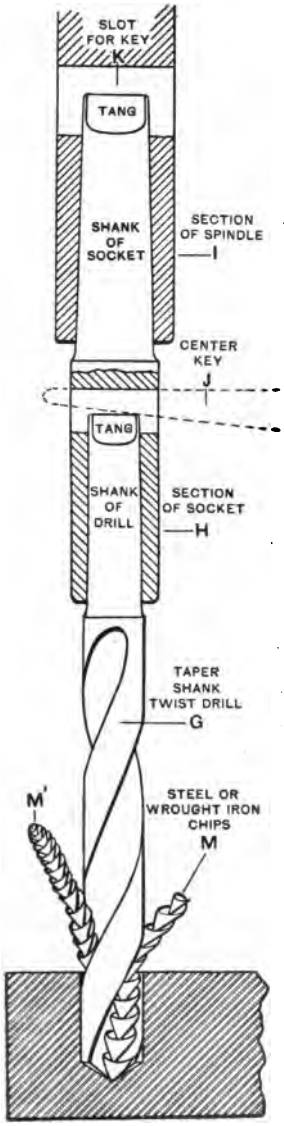


FIG. 289.— DRILLING STEEL OR WROUGHT IRON.

will come out of the hole in long shavings if the drill is properly ground and lubricated with lard oil otherwise the chips will break into small fragments.

Attention. — The taper shanks of socket and drill must be clean and forced to their seats in spindle and socket with a quick movement of the hand or lightly driven in with a soft hammer.

Note. — Drills are obtainable with shanks to fit special machines, as blacksmiths' drill presses, etc.

444. A twist drill hot-forged and twisted from a flat bar is shown in

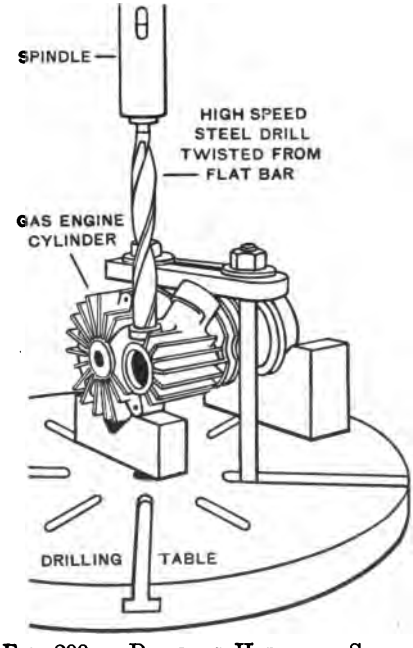


FIG. 290. — DRILLING HOLE FOR SPARK PLUG.

Digitized by Google

Fig. 290 drilling a hole for a spark plug in a gas-engine cylinder. These drills are obtainable to fit both the regular and special sockets and they are made of high-speed steel, and are particularly adapted to rough work of cast iron, boiler-plate work, rail work, or any hard and tough material. Twist drills regular and special may be used on wood by running them at a high speed.

445. A flat drill and set screw drill chuck. — Flat drill *A B*, Fig. 291, is forged from carbon or high-speed steel, having a diameter that will fit set screw drill chuck *C*, and is secured to chuck by set screw *D*.

Forge it a little large at *E*, then grind to size and clearance at *F* and *F'*. Form the cutting lips by grinding.

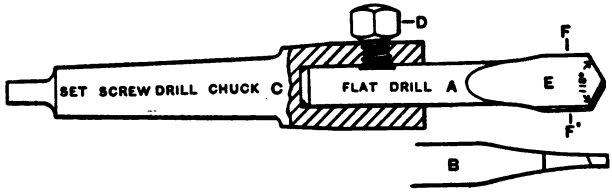


FIG. 291. — FLAT DRILL AND SET SCREW DRILL CHUCK.

This drill is used for all metals, but cuts much slower than a twist drill. It can be made very hard and used for drilling very hard steel that cannot be drilled with a twist drill.

A drill often glazes the surface of hard steel, and will not cut unless the surface is roughed up by indentations with a chisel. See Drilling Extra Hard Steel, § 101.

FIG. 292. — TWIST DRILL GROUND FOR BRASS OR SHEET METAL.

446. To grind a drill for brass or thin sheet metal. — The helical grooves of a twist drill give the cutting lips acute cutting angles which are suitable for steel, wrought iron, and cast iron. To drill brass or thin sheet metal, grind top faces of both lips flat, about $\frac{1}{32}''$, to remove the rake, as at *A*, Fig. 292, otherwise the drill is liable to "dig in," catch and break.

Digitized by Google

447. A straight-groove drill, *A*, Fig. 293, has straight grooves which make it a safe drill for any of the compositions

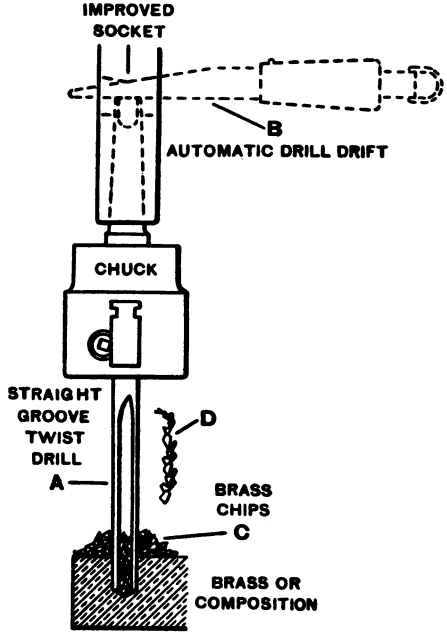


FIG. 293. — DRILLING BRASS OR COMPOSITION.

of brass or thin sheet metal. It is grooved the same as a twist drill. The chuck (socket or drill) may be removed by inserting the automatic key or drift *B* in the slot with the right hand and striking it a blow with its weighted handle operated with the right hand and receiving the chuck with the left. It acts as a combination drift and hammer. These drills are obtainable in parts of an inch and also in wire gage sizes.

448. Drilling brass or composition, Fig. 293. — The chips will come out of the hole in small angular fragments as at *C*, if the brass or composition is hard (cast); and in long shavings as at *D*, if soft (rolled).

Digitized by Google

SIZES OF DRILLS.

449. Regular twist drills with straight or taper shanks are obtainable in binary fractions, mixed numbers, and whole number sizes from $\frac{1}{16}''$ to 3″, and by special order from $3\frac{1}{16}''$ to 6″, and for further convenience, in various gage sizes for pinning, doweling, selecting taps, reamer drills, etc. See §§ 450, 451, 452. Left-hand drills are obtainable from $\frac{1}{16}''$ to $\frac{1}{2}''$ by $\frac{1}{64}''$. Drills are also obtainable in metric sizes.

Attention. — Holes up to 3″ are usually drilled or drilled and reamed, holes from 3″ to 6″ drilled or bored. Holes above 6″ are bored.

Note. — In holes larger than two inches it is best to drill a hole from one-half to two-thirds the required size with a two-groove drill and finish it with a three or four-groove drill. See § 478.

450. Large drills are selected by the sizes stamped on shanks or they may be measured with calipers. In regular drills, one-half inch and less, gages and tables of decimal equivalents are obtainable. See Fig. 294 and Table.

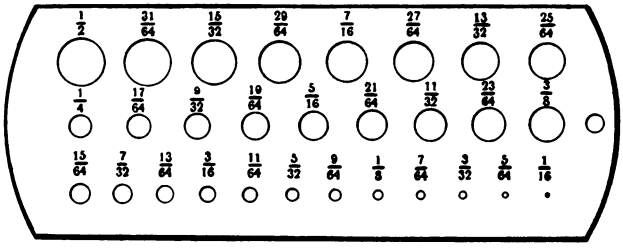


FIG. 294. — GAGE FOR FRACTIONAL SIZE DRILLS.

TABLE OF GAGE SIZES IN COMMON FRACTIONS AND DECIMAL EQUIVALENTS.

SIZE.	DECIMAL.	SIZE.	DECIMAL.	SIZE.	DECIMAL.	SIZE.	DECIMAL.	SIZE.	DECIMAL.
$\frac{1}{16}$	.0625	$\frac{5}{32}$	.15625	$\frac{1}{4}$	.25	$\frac{11}{32}$	.34375	$\frac{7}{16}$	.4375
$\frac{5}{64}$	.07812	$\frac{11}{64}$	.17187	$\frac{17}{64}$	.26562	$\frac{23}{64}$	.35937	$\frac{29}{64}$	.45312
$\frac{3}{32}$	.09375	$\frac{3}{16}$	.1875	$\frac{9}{32}$	.28125	$\frac{3}{8}$	.375	$\frac{15}{32}$	.46875
$\frac{7}{64}$	.10937	$\frac{13}{64}$	.20312	$\frac{19}{64}$	.29687	$\frac{25}{64}$	.39062	$\frac{31}{64}$	.48437
$\frac{1}{8}$	.125	$\frac{7}{32}$	.21875	$\frac{5}{16}$	.3125	$\frac{13}{32}$	.40625	$\frac{1}{2}$	.50
$\frac{9}{64}$	.14062	$\frac{15}{64}$	.23437	$\frac{21}{64}$	.32812	$\frac{27}{64}$	.42187		

Digitized by Google

451. Letter Drills. — For a greater variety of sizes of drills than those already mentioned, 26 drills are obtainable with a difference between consecutive sizes of from four thousandths to fourteen thousandths and designated by letters from A to Z.

TABLE OF LETTER-SIZE DRILLS AND EQUIVALENTS IN DECIMALS AND COMMON FRACTIONS.

Letter Sizes.	Decimals of an Inch.	Nearest 64th of an Inch.	Letter Sizes.	Decimals of an Inch.	Nearest 64th of an Inch.
A	.234	$\frac{15}{64}$	N	.302	.....
B	.238	.....	O	.316	$\frac{5}{16}$
C	.242	.....	P	.323	$\frac{21}{64}$
D	.246	.....	Q	.332	.....
E	.250	$\frac{1}{4}$	R	.339	$\frac{11}{32}$
F	.257	.....	S	.348	.....
G	.261	.....	T	.358	$\frac{23}{64}$
H	.266	$\frac{17}{64}$	U	.368	.....
I	.272	.....	V	.377	$\frac{3}{8}$
J	.277	.....	W	.386	$\frac{25}{64}$
K	.281	$\frac{9}{32}$	X	.397	.....
L	.290	.....	Y	.404	$\frac{13}{32}$
M	.295	$\frac{19}{64}$	Z	.413	.....

452. Small drills are also obtainable in sets designated by numbers from 1 to 80. These drills have a range of from one to eight thousandths of an inch between consecutive sizes.

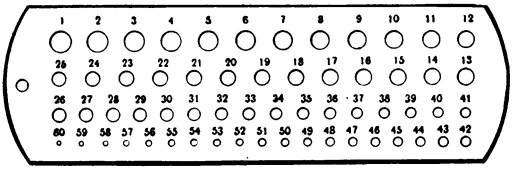


Fig. 295. — Twist Drill and Steel Wire Gage. Sizes by Numbers, 1 to 60. Half Size.

See Gages, Figs. 295 and 296, Tables of Decimal Equivalents, and American Screw Company's Machine and Wood Screws, § 656.

Digitized by Google

TABLE OF SIZES OF TWIST DRILLS AND STEEL WIRE GAGE IN DECIMALS OF AN INCH, 1 TO 60.

No.	Size of Number.	No.	Size of Number.	No.	Size of Number.	No.	Size of Number.
1	.2280	16	.1770	31	.1200	46	.0810
2	.2210	17	.1730	32	.1160	47	.0785
3	.2130	18	.1695	33	.1130	48	.0760
4	.2090	19	.1660	34	.1110	49	.0730
5	.2055	20	.1610	35	.1100	50	.0700
6	.2040	21	.1590	36	.1065	51	.0670
7	.2010	22	.1570	37	.1040	52	.0635
8	.1990	23	.1540	38	.1015	53	.0595
9	.1960	24	.1520	39	.0995	54	.0550
10	.1935	25	.1495	40	.0980	55	.0520
11	.1910	26	.1470	41	.0960	56	.0465
12	.1890	27	.1440	42	.0935	57	.0430
13	.1850	28	.1405	43	.0890	58	.0420
14	.1820	29	.1360	44	.0860	59	.0410
15	.1800	30	.1285	45	.0820	60	.0400

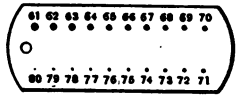


Fig. 296. — Twist Drill and Steel Wire Gage. Sizes by Numbers, 61 to 80. Half Size.

TABLE OF SIZES OF TWIST DRILL AND STEEL WIRE GAGE IN DECIMALS OF AN INCH, 61 TO 80.

Number.	Decimals.	Number.	Decimals.	Number.	Decimals.	Number.	Decimals.
61	.039	66	.033	71	.026	76	.02
62	.038	67	.032	72	.025	77	.018
63	.037	68	.031	73	.024	78	.016
64	.036	69	.02925	74	.0225	79	.0145
65	.035	70	.028	75	.021	80	.0135

453. There are three classes of drills: body, reamer, and tap drills.

Body drills are used to drill a hole to a nominal diameter as stamped on the drill, — a $\frac{3}{4}''$ drill for a $\frac{3}{4}''$ hole.

Digitized by Google

Reamer drills are used to drill a hole that is to be finished by reaming and are usually from $\frac{1}{100}$″ to $\frac{1}{64}$″ smaller than the reamer.

Tap drills are used to drill for tapping and have a diameter equal to that of the tap at root of thread plus the clearance. See Tapping and Threading Sizes, § 534, and also Tables of Tap Drills, §§ 538, 569, 570, 573, 574.

SPEEDS AND FEEDS OF DRILLS.

454. To calculate revolution of drill (R.P.M.) given the cutting speed and the diameter of drill.

Multiply the cutting speed by 12 to reduce it to inches, then divide by the diameter of the drill multiplied by 3.1416.

Example. — The drill is $\frac{3}{4}$″ in diameter (high-speed steel drill) and it is desired to drill at 50 F.P.M. How many drill revolutions are necessary?

Solution. —

$$\frac{50 \times 12}{.750 \times 3.1416} = 254 \text{ R.P.M.}$$

455. To calculate cutting feet of drill (F.P.M.) given diameter of drill and drill revolutions per minute.

Multiply diameter of drill by 3.1416 and by the number of drill revolutions, then divide by 12 to reduce to feet.

Example. — The drill is $\frac{3}{4}$″ in diameter and makes 254 revolutions per minute. What is the cutting speed?

Solution. —

$$\frac{.750 \times 3.1416 \times 254}{12} = 50 \text{ F.P.M.}$$

456. Table of carbon-steel drill speeds and feeds, giving both R.P.M. and F.P.M. in order that speed may be tested with a speed indicator. See *Elements of Machine Work*, or Cut-Meter, § 94.

Feed is given in thousandths of an inch per revolution of drill.

Digitized by Google

Diam. of Drill.	Cast Iron.		Machine Steel or Wrought Iron.		Brass.		Diam. of Drill.
	R.P.M. For 35 F.P.M.	Feed P.R.	R.P.M. For 30 F.P.M.	Feed P.R.	R.P.M. For 70 F.P.M.	Feed P.R.	
1/16	2138	.002	1832	.002	4276	.002	1/16
1/8	1070	.002	916	.002	2138	.002	1/8
3/16	712	.002	610	.002	1424	.002	3/16
1/4	535	.004	458	.004	1070	.004	1/4
5/16	427	.004	366	.004	854	.004	5/16
3/8	356	.004	305	.004	712	.004	3/8
7/16	305	.004	261	.004	610	.004	7/16
1/2	267	.008	229	.006	534	.008	1/2
	For 30 F.P.M.		For 25 F.P.M.		For 60 F.P.M.		
5/8	183	.008	152	.006	366	.008	5/8
3/4	134	.008	127	.006	268	.008	3/4
7/8	130	.008	108	.006	260	.008	7/8
1	114	.012	95	.009	228	.012	1
1 1/4	90	.012	76	.009	180	.012	1 1/4
1 1/2	67	.012	63	.009	134	.012	1 1/2
	For 25 F.P.M.		For 20 F.P.M.		For 50 F.P.M.		
1 3/4	54	.012	43	.009	108	.012	1 3/4
2	47	.016	38	.012	94	.016	2
2 1/4	42	.016	33	.012	84	.016	2 1/4
2 1/2	38	.016	30	.012	76	.016	2 1/2
2 3/4	34	.016	27	.012	68	.016	2 3/4
3	32	.016	25	012	64	.016	3

Attention. — Carbon steel varies in hardness. When annealed it is moderately soft and may be drilled at nearly as high a speed as machine steel, but unannealed it is hard and will require a reduction to about one-half the speed of machine steel.

Note. — If the cutting of the drill and chips indicates that the metal is soft, the speed and feed may be increased, and vice versa if the metal is hard. Keep drills sharp. If a drill is run too fast, the outer ends of cutting lips will dull quickly and wear away, as a high speed will generate excessive heat and draw the temper at these points first. If the feed is too coarse, the drill will chip at the cutting edges, or break, or the drilling machine will stop.

Digitized by Google

457. Table of high-speed steel drill speeds and feeds, giving both R.P.M. and F.P.M. in order that speed may be tested with a speed indicator (see *Elements of Machine Work*), or Cut-Meter, § 94.

Feed is given in thousandths of an inch per revolution of drill.

Diam. of Drill.	Cast Iron.		Steel and Wrought Iron.		Diam. of Drill.
	R.P.M. For 70 F.P.M.	Feed P.R.	R.P.M. For 60 F.P.M.	Feed P.R.	
1/16	4276	.003	3664	.002	1/16
1/8	2138	.003	1832	.002	1/8
3/16	1424	.003	1220	.002	3/16
1/4	1070	.006	916	.004	1/4
5/16	854	.006	732	.004	5/16
3/8	712	.006	610	.004	3/8
7/16	610	.009	522	.008	7/16
1/2	534	.009	458	.008	1/2
	For 60 F.P.M.		For 50 F.P.M.		
5/8	366	.013	304	.012	5/8
3/4	268	.013	254	.012	1/4
7/8	260	.013	216	.012	7/8
1	228	.013	190	.012	1
1 1/4	180	.018	152	.016	1 1/4
1 1/2	134	.018	126	.016	1 1/2
	For 50 F.P.M.		For 40 F.P.M.		
1 3/4	108	.018	86	.016	1 3/4
2	94	.027	76	.020	2
2 1/4	84	.027	66	.020	2 1/4
2 1/2	76	.027	60	.020	2 1/2
2 3/4	68	.027	54	.020	2 3/4
3	64	.027	50	.020	3

Attention. — Speed for brass twice the speed for cast iron, and same feed per revolution.

458. Tables of spindle speeds (R.P.M.) for belt-driven vertical drilling machine and speed lathe. — Select the R.P.M.

Digitized by Google

from the Tables of Drill Speeds and Feeds, etc., § 456 or § 457, and place belt on step of cone that gives the nearest

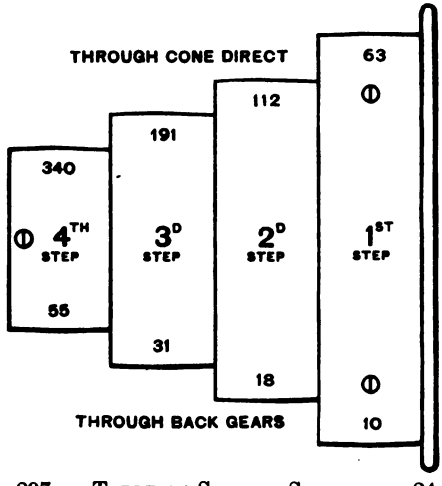


FIG. 297. — TABLE OF SPINDLE SPEEDS FOR 24-INCH VERTICAL DRILLING MACHINE.

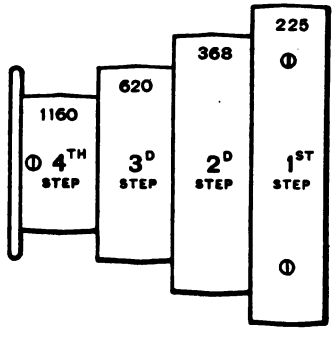


FIG. 298. — TABLE OF SPINDLE SPEED FOR 11-INCH SPEED LATHE.

R.P.M. obtainable with machine as given in Tables, Figs. 297 and 298.

459. Hand and power feeds. — Vertical drilling machines with belt feed usually have three feeds varying, with the different machines, from .002″ to .005″ for a fine feed, or from

Digitized by Google

.007″ to .010″ for a medium feed, and from .015″ to .120″ for a coarse feed, per revolution.

Belt spindle driven machines, with gear feed boxes, give a range of feeds from .004″ to .039″ per revolution of spindle. Use feed per revolution of spindle that is nearest to that given in Tables of Drills, Speeds and Feeds, §§ 456, 457.

Speed lathes and most of the small vertical and horizontal drilling machines have hand feed only, obtainable by a hand wheel or a lever. After grinding a large drill, it is good practice before applying the power feed to start it by hand to see if it is cutting correctly as determined by the chips produced. Very small drills should be started and may be fed by hand to avoid breaking.

460. Table of spindle speeds for a standard high-speed drilling machine which gives the positions of the levers in the speed gear box for 16 speeds at 70 F.P.M. for drilling or boring holes from ½″ to 7″ in diameter.

Hole	½″	⅝″	¾″	7/8″	Without
Revolutions	534	428	356	306	backgear.
Hole	1″	1¼″	1½″	1¾″	With lower
Revolutions	267	214	178	153	backgear.
Hole	2″	2½″	3″	3½″	With upper
Revolutions	134	107	89	76	backgear.
Hole	4″	5″	6″	7″	With both
Revolutions	67	53	45	38	backgears.

461. Table of gear feeds for a high-speed drilling machine obtained from gear feed box in thousandths of an inch per revolution of spindle.

.006″	.009″	.013″
.018″	.027″	.039″

Digitized by Google

LAYING OUT AND DRILLING HOLES.

462. To drill, ream, tap, and counterbore large or medium-size work, use a vertical drilling machine and clamp work to table or hold it by the aid of a vise, angle plate, jig, or fixture, in order that the setting of the work may not be disturbed during operations, for if disturbed the accuracy may be lost, and in many cases the work spoiled. Very large work may be clamped to the base table of the machine. Very often it is more convenient to drill small work in a speed lathe. See §§ 489–491.

463. To set spring dividers, *A*, Fig. 299, to an accurate dimension, place lower point in a line on rule *B*, as at *C*, and move upper point by nut *D* to coincide with line *E*. At *A'*, *B'* is shown how rule is tilted so that dividers may be set accurately.

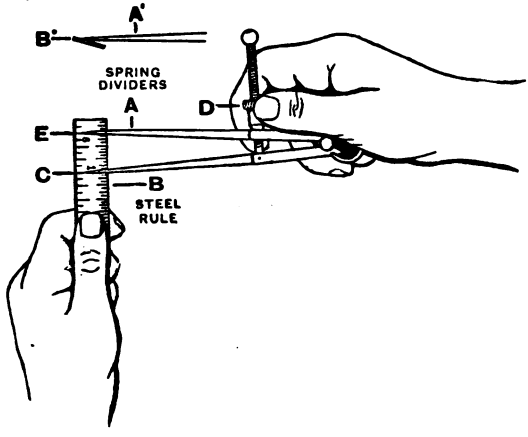


FIG. 299. — SETTING DIVIDERS TO A STEEL RULE.

464. To use dividers to describe a circle, Fig. 300. — Set dividers *F* to size, place block *G* in vise *H* and indent with light center-punch mark at desired location. Then place point *K* in mark, incline dividers slightly in direction of revolution and describe circle with point *L*.

Digitized by Google

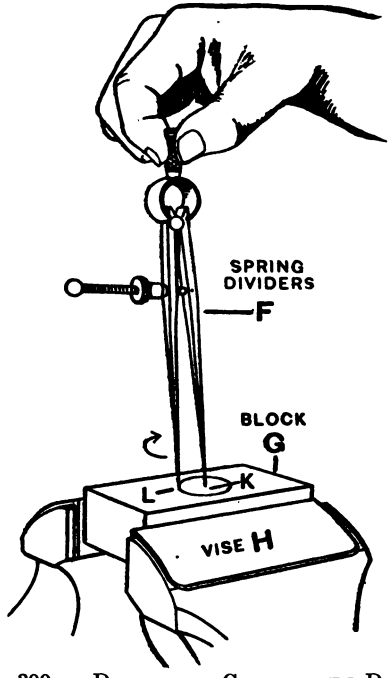


FIG. 300. — DESCRIBING CIRCLE WITH DIVIDERS.

465. Rough drilling. — Determine location of hole in work by the eye, then make a heavy center-punch mark in which to start point of drill.

466. Approximate drilling. — Use dividers or surface gage to locate the hole, then make heavy center-punch mark in which to start drill. Drill hole without drawing the drill.

467. Accurate drilling, laying out work, Fig. 301.

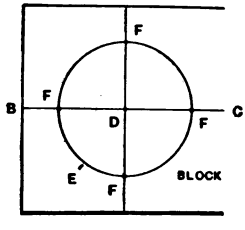


FIG. 301. — LAYING OUT WORK FOR ACCURATE DRILLING.

Digitized by Google

SCHEDULE OF OPERATIONS.

1. Brighten surface of block and coat with copper sulphate. See *Elements of Machine Work.*

2. Draw center line *BC* and intersecting line *FDF* lightly with surface gage, and center punch at point of intersection *D*.

3. Describe circle *E* diameter of drill and make four or more light center-punch marks at *F*.

4. Enlarge center-punch mark *D* to guide drill in starting.

Attention. — For large holes, additional circles may be described inside of size circle to determine accuracy of cut before it reaches the size circle.

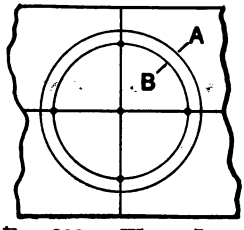


FIG. 302.— WORK LAID OUT WITH AN INSPECTION CIRCLE.

468. Inspection circle. — Work that is laid out by one person and drilled by another should have an inspection circle, as at *A*, Fig. 302, outside the size circle *B*. This is an inspection check on both the laying out and the drilling.

469. Drawing the drill, Fig. 303.

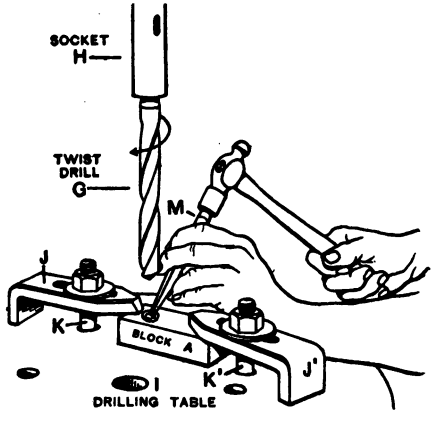


FIG. 303. — CUTTING A GROOVE WITH CENTER CHISEL TO DRAW THE DRILL.

Digitized by Google

SCHEDULE OF OPERATIONS.

1. Insert drill *G* in socket *H*. Swing table *I* until drill will pass through any convenient hole in table. Clamp and clean table.

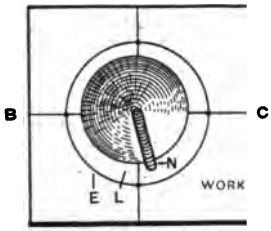


FIG. 304. — GROOVE TO DRAW THE DRILL.

2. Place block *A* on table and bring drill down into center punch hole. Clamp with *JJ'* and *KK'*.

3. Start drill and feed by hand to cut a little; if not cutting concentrically as at *L*, Fig. 304, raise drill and "draw" hole with center chisel *M*, Fig. 303, by cutting groove *N*, Fig. 304.

For a small amount, cut as at *N'*, Fig. 305.

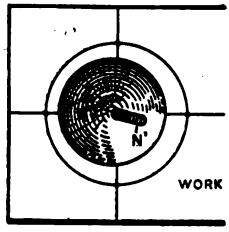


FIG. 305. — GROOVE TO DRAW A DRILL A SMALL AMOUNT.

4. Start drill again, using hand-feed; if not cutting centrally, repeat 3.

5. Loosen work and bring drill down into clean conical hole, reclamp, and finish hole with power feed.

Attention. — A drill cannot be "drawn" after it has cut to its full diameter.

In starting a thick-pointed drill, particularly if a light pressure is used, it may cut an irregular cavity, as at *A*, Fig. 306, which may be drawn as at *B* in the same manner as a true cavity.

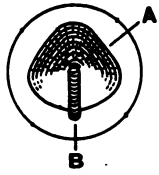


FIG. 306. — IRREGULAR CAVITY CUT BY DRILL.

Digitized by Google

470. Accurate drilling, reaming, and tapping holes I, II, III, Fig. 307.

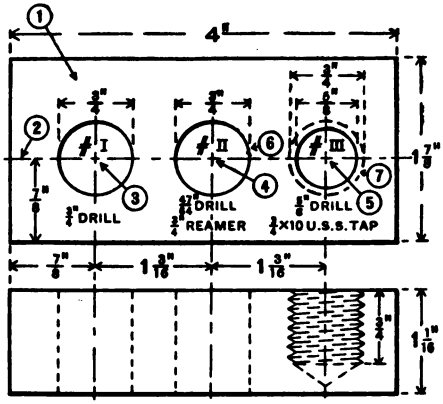


FIG. 307. — SCHEDULE DRAWING OF ACCURATE DRILLING, REAMING, AND TAPPING.

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.

Stock, cast iron, planed and free from visible defects.
Oil bearings of drilling machine with machine oil.
Set hole in table in alinement with drill.
Use carbon-steel drills.
For High-speed Drills, see § 457.

OPERATIONS.	MACHINES, SPEEDS, FEEDS.	TOOLS.
Prepare surface for marking. (1) See *Elements of Machine Work*.	Vise.	Copper jaws, 8″ or 10″ hand smooth file, copper sulphate.
Draw line (2), intersect at (3), (4), (5), center punch intersections.		Level plate, surface gage or dividers, rule, automatic center punch. (Magnifying glass optional.)
Describe circles exact diameter of drill ($\frac{1}{4}$, $\frac{47}{64}$, $\frac{5}{8}$), center punch four intersections on circle.		Dividers, rule, automatic center punch.
Enlarge centers, (3), (4), (5).		Hammer, center punch.

Digitized by Google

TO DRILL HOLE NO. I.

Insert drill in spindle and clamp block.	Vertical drilling machine.	$\frac{3}{4}$″ taper shank, twist drill, wrench, clamp bolts and nuts.
Start drill in center, (3).	2d or 3d speed, or 170 R.P.M. Hand feed.	
Draw drill if necessary.	.....	Hammer, center chisel.
Loosen clamp, center block (see § 469), reclamp and drill hole.	2d or 3d speed, or 170 R.P.M. Power feed.	Wrench.

TO DRILL AND REAM HOLE NO. II.

Drill same as hole No. I, (4).	Vertical drilling machine.	$\frac{47}{64}$″ drill, $\frac{3}{4}$″ reamer drill.
Replace drill by a center.	.....	Conical center.
Ream hole (see § 290), (6).	.....	$\frac{3}{4}$″ hand reamer, reamer wrench.

TO DRILL AND TAP BOTTOM HOLE NO. III.

Drill same as hole No. I ($\frac{3}{4}$″ depth), (5).	Vertical drilling machine. 2d or 3d speed, or 225 R.P.M.	$\frac{5}{8}$″ drill, depth gage.
Place block in vise and tap hole (see § 527), (7).	Vise.	Copper jaws, $\frac{3}{4}$″ × 10 United States Standard taper tap, tap wrench, sperm oil.
Use plug tap. Use bottom tap.	.....	$\frac{3}{4}$″ × 10 United States Standard plug tap $\frac{3}{4}$″ × 10 United States Standard bottom tap.

471. To drill for set screws. — Locate and drill with a tap drill, then tap the hole. See Set Screws, § 563.

Digitized by Google

472. Allowance or limit for bolt holes. — For rough work, locate and drill holes with a drill $\frac{1}{64}''$ or more, large, to give the bolts body clearances as at *A*, Fig. 308.

On fine work the holes are drilled with a body drill, or drilled with a reamer drill and reamed, and the bolts fitted to the holes.

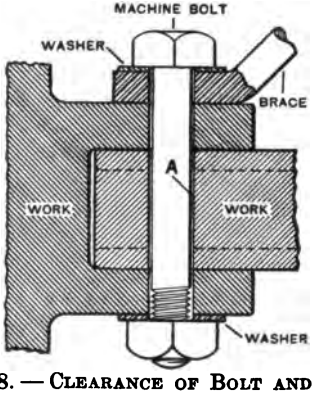


FIG. 308. — CLEARANCE OF BOLT AND WORK.

473. Allowance or limit for cap screws or stud bolts. — For rough work drill a hole in cap with a drill $\frac{1}{100}''$ or more, large, for body clearance, as at *A*, Fig. 309, and a body drill for fine work, then drill hole in frame or bottom part of work with a tap drill and tap, as at *B*.

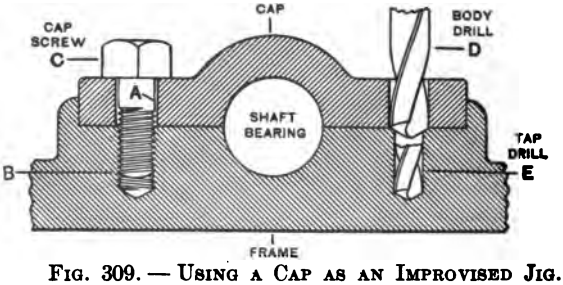


FIG. 309. — USING A CAP AS AN IMPROVISED JIG.

474. To drill in alinement for cap screws without jig. Three methods. —

First Method. — Locate and drill a body hole in cap, then put cap in place and use it as a templet to mark circle *A*,

Digitized by Google

Fig. 310. Make four center-punch marks *B*, and with dividers locate center of circle and make center-punch mark *C*. With *C* as center, describe circle *D*, diameter of tap drill. See Accurate Drilling, §§ 467–471.

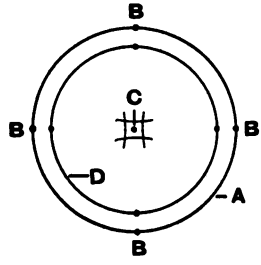


FIG. 310. — LOCATING CENTER OF TAP HOLE.

Second Method. — Locate and drill the body hole in cap, clamp it to bottom part and use it as a jig. Run body drill down hole in cap and make it cut its full diameter in base only. Then remove and use the tap drill to drill the tap hole. When bolting a cap to a shaft bearing or pillow block in which two or more cap screws are used, it is best to drill and tap one hole and use a cap screw, *C*, Fig. 309, to secure the cap to the shaft bearing while drilling the other holes, *D* and *E*, Fig. 309.

Attention. — If the work has a large number of cap screws or studs, as a steam-chest cover or cylinder head, and no jig is provided, a tap drill bushing may be used in the body hole in cap to guide the tap drill accurately. See Jigs, §§ 495–504.

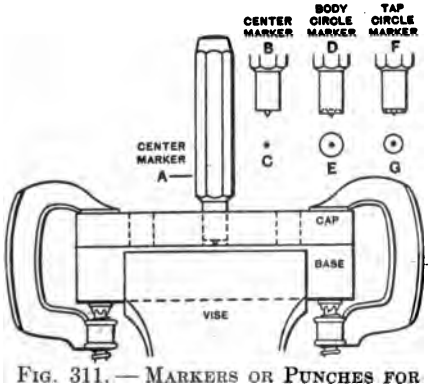


FIG. 311. — MARKERS OR PUNCHES FOR LAYING OUT DUPLICATE WORK.

Third Method.—Clamp cap to pillow block, or clamp any two pieces together, locate and drill both parts with tap drill, then take cap or top part off and enlarge its hole to body size of screw with a counterbore whose head diameter is the same as the body of the screw, and with a guide the diameter of tap drill. See Counterboring, § 575.

475. Markers for laying out duplicate work, Fig. 311. — Plates to be drilled may be laid out rapidly by using one

Digitized by Google

as a templet, as the cap, Fig. 311, then using a marker, as at *A*. Other styles of marker are shown at *C*, *E*, and *G*.

476. To drill work held by hand or in a vise on drilling table, Fig. 312. — Long work is often held on drilling table by hand, or a bolt may be inserted in a hole in the table to prevent work revolving with the drill. Small square or flat pieces may be held by a monkey wrench, see Fig. 315. A safer and more accurate method is to hold work in a vise as in Fig. 312. Clean table, jaws, and base of vise. Clamp vise *A* to table *B* by two bolts, one of which is shown at *C*, to hold block *D* to be drilled with drill *E* held in chuck *F*.

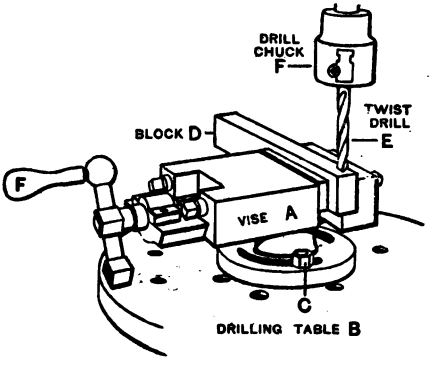


FIG. 312. — DRILLING WORK HELD IN VISE. VERTICAL DRILLING MACHINE.

Place work against fixed jaws and bring the sliding jaw up by handle. Rap the work lightly with a lead hammer to bed it properly, during and after closing of vise.

477. To drill work held against an angle plate. See § 278, Fig. 313. — Bolt angle plate *A* to table *B* by bolts, as at *C*. Clamp bracket *D* to vertical face of angle plate by straps and bolts *E* and *E'*. Drill and ream hole through part *F* parallel to base. Jack screw *G* supports work.

478. Three and four-groove twist drills for drilling cored holes or to follow a two-groove twist drill. Four-groove twist drill *H*, Fig. 313, is used to drill or enlarge cored hole in

Digitized by Google

bracket casting *D*. These drills are usually made $\frac{1}{64}$″ small and are followed by a standard shell reamer.

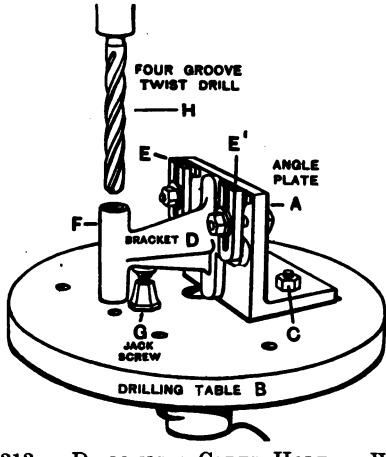


FIG. 313. — DRILLING A CORED HOLE IN WORK CLAMPED TO ANGLE PLATE.

479. To drill diametrically through a shaft, Fig. 314. — Secure shaft *A* on drilling table *B* in V-groove blocks *C* and

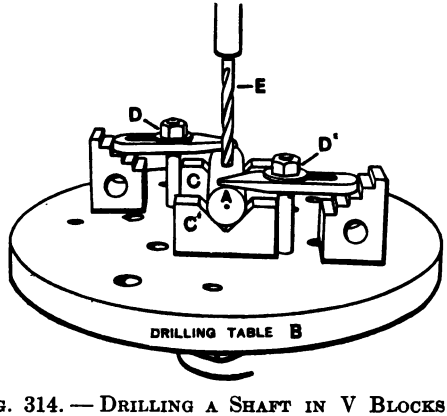


FIG. 314. — DRILLING A SHAFT IN V BLOCKS. VERTICAL DRILLING MACHINE.

C' by clamps and bolts *D* and *D'*. Lay out shaft *E* in the usual way, and set central as follows: Place try square upon

Digitized by Google

table with the edge of the blade against shaft, first at one side, then at the other, measuring distance from the blade to the center mark and adjusting. Then move table until the point of drill will enter center-punch mark, clamp and drill hole.

480. Automatic drill chuck and collet, Fig. 315.

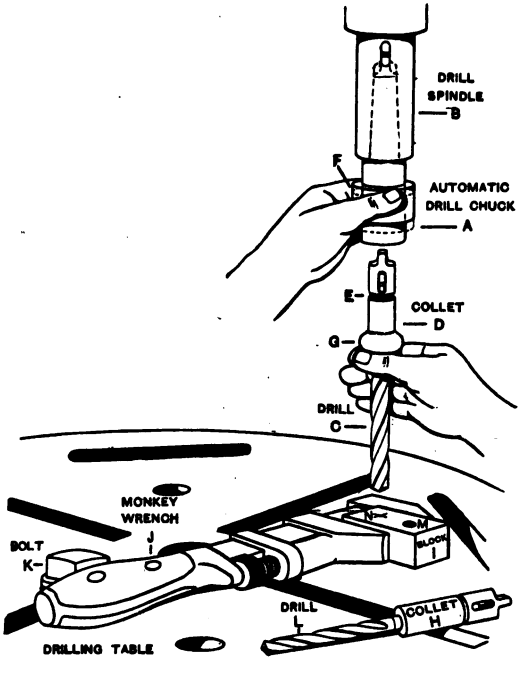


FIG. 315. — CHANGING DRILLS WITHOUT STOPPING MACHINE BY USING AN AUTOMATIC CHUCK AND COLLET.

SCHEDULE OF PARTS AND OPERATIONS.

A — Automatic drill chuck taper shank, fits spindle.	*E* — Groove.
B — Drill press spindle.	*F* — Locking ring. Raise it, insert collet, then drop it, which will lock two pins in chuck *A* into groove *E*. To remove collet, raise ring and it will drop out.
C — Taper shank drill, fits taper hole in collet.	
D — Collet, straight shank.	

Digitized by Google

SCHEDULE OF PARTS AND OPERATIONS. *Concluded.*

G — Spherical end of collet which prevents it from slipping through hand.

H — Collet made with straight end for use with small drills.

I — Block; requires two holes drilled.

J — Wrench to hold block.

K — Bolt, inserted to prevent wrench from revolving.

L — Drill for first hole.

M — First hole drilled, after which drill is removed without stopping machine.

N — Second hole to be drilled. Drill *C* inserted as shown, without stopping machine.

The collet is driven by its tang and held in place by the locking pins.

481. Cutting metal plates by drilling. — A metal plate may be cut apart, if the cut is straight, by planing, milling, or shearing; but if the cut is curved, as in Fig. 316, drill holes *A*, *B*, *C*, etc., about 1½ diameters apart, and sever walls *D*, *E*, *F*, etc., with chisel.

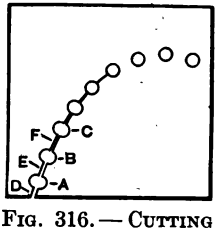


FIG. 316. — CUTTING A METAL PLATE BY DRILLING AND CHIPPING.

482. Slotting by drilling, plugging, and drilling. — The slot is laid out on both sides, Fig. 317. As a hole that cuts into another cannot be drilled, drill holes *A*, *B* and *C* apart, then plug and drill holes *D* and *E* to cut away the walls. Drive out plugs and finish slot by filing, slotting, or milling. As a time saver for large lots, a jig may be made.

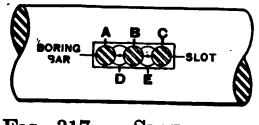


FIG. 317. — SLOTTING BY DRILLING, PLUGGING, AND DRILLING.

483. Lead holes for large drills, Fig. 318. To make the large drill *A* cut faster, first drill "lead" hole *B* (diameter of drill same as thickness of point or larger) through work *C*. Draw drill with center chisel as at *D*, Fig. 319.

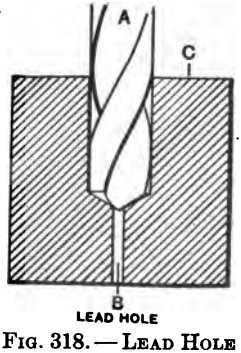


FIG. 318. — LEAD HOLE FOR A LARGE DRILL.

Digitized by Google

484. To start drill when the surface is slanting, Fig. 320. — First chip or mill surface perpendicular to the drill, as shown at *A*. Use chalk or copper sulphate on this surface. Center and lay out with dividers, then drill in the regular way.

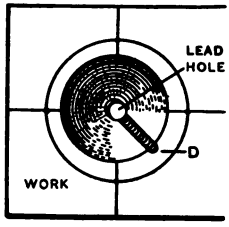


FIG. 319.— DRAWING A DRILL WHERE A LEAD HOLE IS USED.

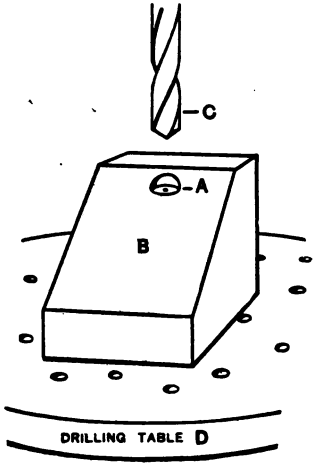


FIG. 320. — DRILLING HOLE ON SLANTING SURFACE.

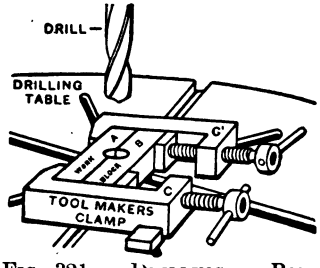


FIG. 321. — DRILLING A PORTION OF A HOLE IN A PIECE.

485. To drill a part of a hole in the side of a block as at *A*, Fig. 321. — Clamp to the piece to be drilled another block, as at *B*, by clamps as at *C*, *C'*. Center and lay out in regular way. The drilling may be done on drilling table of vertical drilling machine, as shown, or at the speed lathe.

486. A boiler tube hole cutter, *A*, Fig. 322. — For large holes in plate work, special drill *A*, with a pin guide *B* to fit punched hole in boiler head *C* is used. It has two cutting lips which cut a circular groove and remove an annular ring.

Digitized by Google

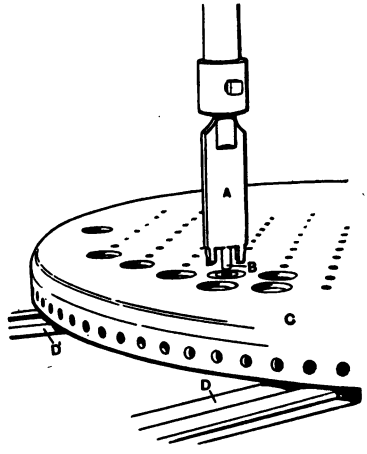


FIG. 322. — DRILLING OR CUTTING HOLES IN BOILER HEAD.

487. Drill extension, Fig. 323. — Make hole *A* (diameter of the drill) in the end of a steel rod of suitable length. File slot *B* across bottom of the drilled hole as at *C*. File flat *D* on the shank of drill. Taper the bottom of slot *B* slightly so that drill will hold firmly in hole *A*. The end of drill should not abut against side *E*.

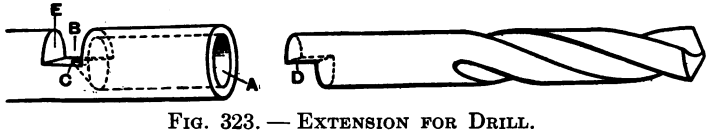


FIG. 323. — EXTENSION FOR DRILL.

488. Leveling work on drilling table. — For rough work, use spirit level. For accurate work, test with a piece of bent wire with an offset and held in chuck. Rotate spindle slowly to swing wire around to test level.

489. To drill small flat or square work in speed lathe, Fig. 324. — Place drill *A* in chuck *B*, which is attached to live spindle by means of an arbor. Hold work *C* against table center *D* by hand vise *E*, and support on Tee rest. When point of drill breaks through, hold work firmly against table

Digitized by Google

center, as there is a tendency for the work to draw away and break the drill.

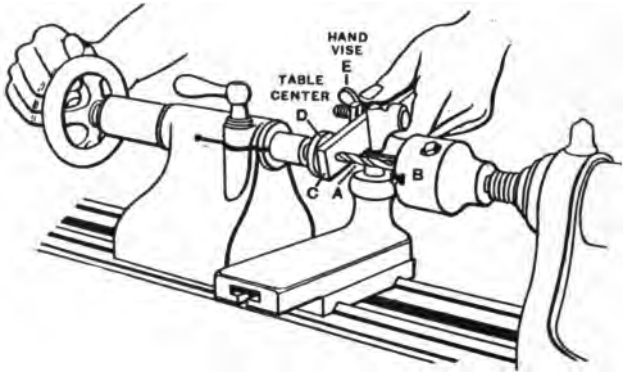


FIG. 324. — DRILLING FLAT WORK IN SPEED LATHE.

490. To drill a hole into or through a shaft at right angles to its axis. — Place shaft *A*, Fig. 325, in V center *B*, and drill *C* in socket *D*, then start drill carefully to make hole central. When hole is to go through work, line out on both sides, drill half-way through from each side, using regular pointed center against work.

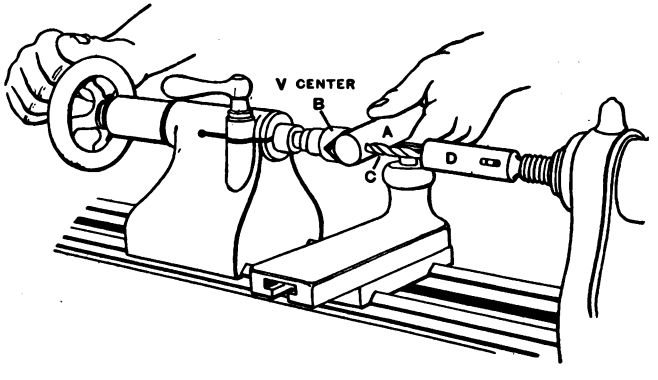


FIG. 325. — DRILLING SHAFT HELD IN V CENTER IN SPEED LATHE.

Digitized by Google

491. To drill a hole in the end of a shaft parallel to axis, Fig. 326. — Place shaft *A* against dead center *B* and drill by

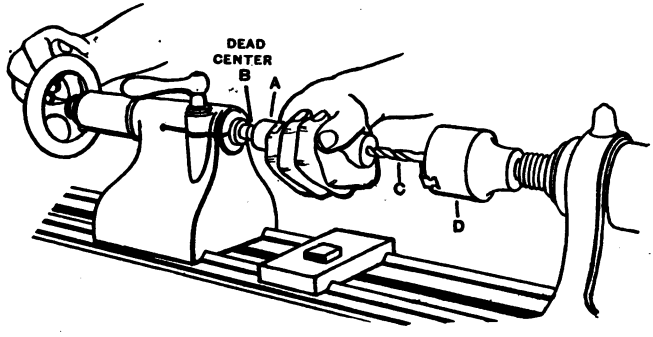


FIG. 326. — DRILLING HOLE IN END OF SHAFT IN SPEED LATHE.

drill *C* in chuck *D*. To drill a hole clear through center of shaft, drill half from each end. See Chucking Work with Steady Rest, *Advanced Machine Work*.

DEEP DRILLING.

492. Oil drills for deep drilling, as at *A*, Fig. 327, provide channels for conveying the oil directly to the point of the drill to keep it cool. These drills are obtainable in sizes from $\frac{3}{8}''$ to 2″ diameter, with both straight and taper shanks.

For chucking, these drills are fitted with a hose-pipe connection which is threaded to the side of the drill.

For rapid drilling, oil is used at a pressure of from 400 lbs. to 600 lbs. to the square inch, not only to lubricate the drill but also to force out the chips.

Attention. — For drilling or chucking in cast iron with these drills, compressed air may be used both to cool the drill and to force the chips out of the drilled hole.

Digitized by Google

493. To drill a deep hole with taper shank oil drill, Fig. 327.

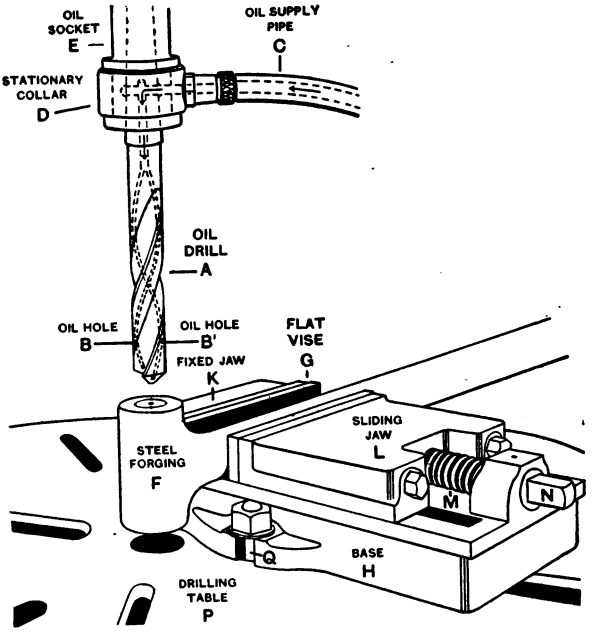


FIG. 327. — DRILLING STEEL FORGING WITH OIL DRILL. VERTICAL DRILLING MACHINE.

SCHEDULE OF PARTS AND OPERATIONS.

A — Oil drill.

B, *B'* — Oil holes through solid metal to point of drill.

C — Oil supply pipe.

D — Stationary collar.

E — Oil socket; oil passes through *C*, hole in *E* to holes *B*, *B'* in drill.

F — Steel forging to be drilled.

G — Flat vise holding forging *F*.

H — Vise base.

K — Fixed jaw.

L — Sliding jaw.

M — Screw bearing against jaw *L*.

N — Square end of screw for wrench to tighten jaws.

P — Drilling table.

Q and *Q'* (*Q'* not shown) — Bolts holding base of vise to table.

Digitized by Google

494. Hollow drills for spindles, tubes, and gun barrels, Fig. 328.

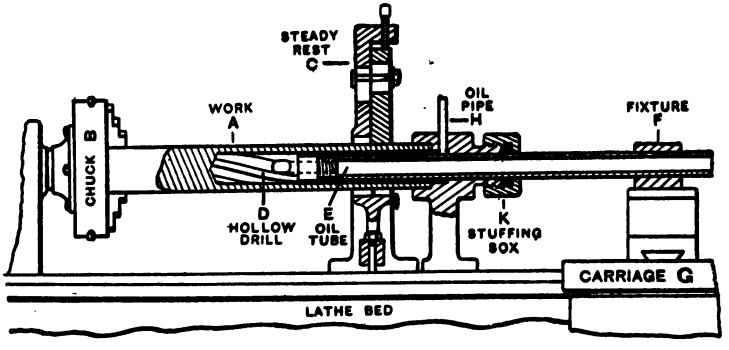


FIG. 328. — DEEP DRILLING WITH HOLLOW OIL DRILL. ENGINE LATHE.

SCHEDULE OF PARTS AND OPERATIONS.

A — Work to be drilled.
B — Chuck.
C — Steady rest.
D — Threaded end of hollow drill screwed into oil tube. Drill shown in detail in Fig. 329.
E — Steel oil tube; oil follows along tube and grooves of drill to the point.
F — Fixture in which hollow drill is clamped.
G — Carriage on which fixture *F* is mounted.
H — Oil pipe.
K — Stuffing box, to prevent oil flowing backward. Oil and chips are forced out through hollow shank of drill *D*.

FIG. 329. — HOLLOW OIL DRILL FOR DEEP DRILLING.

Attention. — To start drill accurately, drill hole in end of work with a regular drill of the same diameter and to a depth equal to the diameter.

Note. — These drills are obtainable from $\frac{5}{8}''$ to $3''$. Deep holes above $3''$ in diameter are drilled or bored with a cutter attached to a boring head which is a running fit in the starting hole. See *Advanced Machine Work.*

Digitized by Google

CHAPTER XVI.

DRILLING JIGS, AND MULTIPLE-SPINDLE DRILLING MACHINES. RADIAL DRILLING MACHINES. TAPS AND DIES.

DRILLING JIGS, AND MULTIPLE-SPINDLE DRILLING MACHINES.

495. The demand for machinery with interchangeable parts has compelled builders to design and use drilling jigs, milling jigs, planing jigs, gages, and templets to produce standard and duplicate parts accurately and economically. See Jig Making, *Advanced Machine Work*.

496. Drilling jigs are fixtures carefully made with hardened bushings to guide drills, reamers, etc., so that their operation shall be the same on each piece. They may be divided into about five classes, and each class is best adapted to some particular kind of work.

1. Plate jigs are used for flanges, machine frames, etc. See §§ 498, 499.

2. Solid jigs are used for work that can be readily clamped to jig body. These jigs are preferred by some for general work. See § 509.

3. Box jigs are used for general work. The work is placed in a box, the hinged cover closed and fastened and the work held in place by binding screws. See §§ 502, 504.

4. Rotary jigs are for work where the jig is too heavy to be easily turned over at the drilling machine. The jig consists of a box mounted on trunnions to facilitate revolution.

5. Multiple jigs are for work that is to be index-drilled. The bushings are placed in a turret head.

497. An improvised jig or templet, Fig. 330. — To drill and ream bolt holes *B*, *C*, and *D*, equidistant, in two or more flanges: First, lay out, drill, and ream one flange carefully. Next, clamp one drilled flange to another in the way they are

Digitized by Google

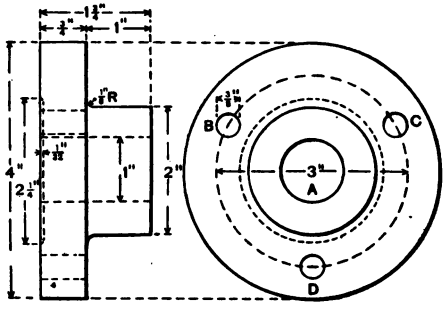


FIG. 330. — COUPLING FLANGE.

to fit, and insert a plug or mandrel through centers *A* to aline them. Drill and ream holes in the other flange. Other methods are to drill both flanges then ream together, or drill and ream in pairs.

498. Plate (flange) drilling jig, Fig. 331, is a cast-iron disk *E* supplied with a plug *FG* to aline flange and jig, a hardened

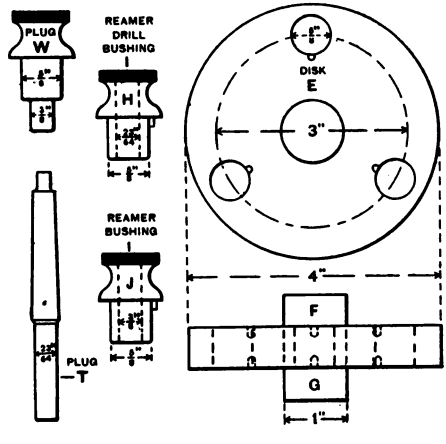


FIG. 331. — FLANGE DRILLING JIG. (PLATE JIG.)

steel drill bushing *H* and reamer bushing *J*. Plug *T* is for alining drill and jig and plug *W* to prevent relative movement of flange and jig.

Digitized by Google

499. To use flange jig, Fig. 332. — Place flange *K* on parallel pieces *L* and *M*, with jig *N* on top of flange; clamp lightly by strap *P*, block *Q*, and bolt *R* to table *S*. Aline jig and drill spindle with plug *T*, Fig. 331, moving work by rapping with a soft hammer until the plug will enter bushing exactly central; clamp firmly. Drill hole with reamer drill *U*. Substi-

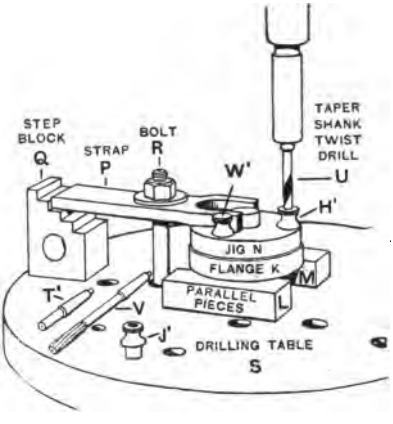


FIG. 332. — DRILLING AND REAMING WITH FLANGE JIG.

tute reamer bushing *J'* for drilling bushing *H'*, and ream with fluted reamer *V*. Place plug *W*, Fig. 331, in first hole to keep jig and flange in alinement as at *W'*, Fig. 332, while drilling and reaming second and third holes. Reverse jig to drill the second flange.

500. Drilling and tapping engine cylinder heads. — When two pieces of work are to be clamped together with cap screws or stud bolts, as a cylinder and cylinder head, two removable bushings are required for the jig; the first with body drill holes for the head, the second with tap drill holes for the cylinder. To tap the holes by hand, a tap bushing is sometimes used to guide the tap. See Automatic Tapping Attachment, Fig. 335.

501. Multiple-spindle drilling machines are used to save time in changing drills, reamers, and counterbores, in moving work

Digitized by Google

from one machine to another, as is necessary with a one-spindle drilling machine. One spindle holds a tap drill, another a body drill, another a counterbore, etc., all running, and the work is moved along the table from one spindle to another without stopping the machine. See Figs. 333 and 337.

502. Box jigs are used to further increase the rapidity and accuracy of drilling, reaming, tapping, and counterboring.

The work is locked in a box provided with accurately machined bearings to rest on the drilling table. The jigs are made heavy and are held in position by their own weight, thus saving the time that would be consumed in clamping and alining.

503. To drill and ream bolt holes in coupling flange with a box jig and a two-spindle high-speed drilling machine, Fig. 333.

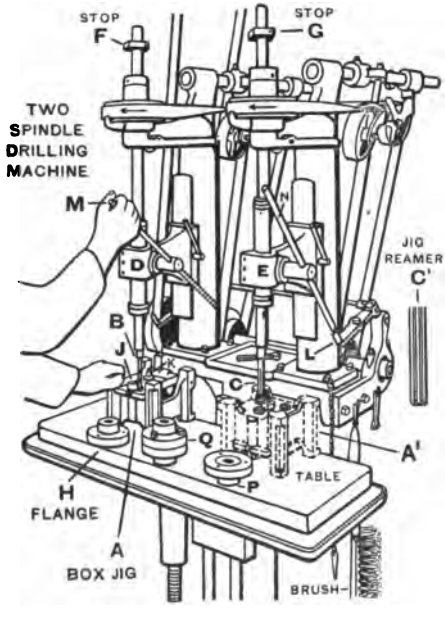


FIG. 333. — DRILLING AND REAMING HOLES IN COUPLING FLANGE WITH BOX JIG.

Digitized by Google

SCHEDULE OF OPERATIONS.

$\frac{1}{64}$″ High-speed steel drill, 600 R.P.M.	$\frac{1}{8}$″ High-speed steel reamer, 400 R.P.M.

1. Place box jig *A* on table. Insert drill *B* and reamer *C*, *C′* in spindles.

Locate heads *D* and *E* on column and clamp stops *F* and *G* on spindles to limit travel of drill and reamer as tested by passing drill and reamer through bushing into empty jig, or test drill and reamer by lines placed on outside of jig which indicate position of bushings.

Place flange *H* in jig with hub up, and secure with button *J* and screw *K*. Start machine by shipper *L*.

2. With left hand move jig to aline drill bushings with drill. Use lever feed *M* and drill three holes.

3. Turn jig over as shown, dotted, at *A′*.

4. Ream the holes, using lever feed *N*. *P* shows position of flange as it is reamed, and *Q* shows two flanges bolted together.

Attention. — Stop *G* must check reamer before it strikes drill bushing or reamer and bushing will be spoiled.

Note. — Before placing jig on table, and before turning jig over, brush off table.

To prevent abrasion of bushings, drill, and reamer, apply a little oil with finger to upper part of drill and reamer.

504. Box jig for pieces to be drilled, reamed, and tapped in different directions, Fig. 334.

SCHEDULE OF PARTS.

A — Box jig for pieces to be drilled, reamed, and tapped in different directions.

B — Duplicate of work held in jig *A*.

C — Cover held in place by thumb nut *D*.

E and *E′* — Two of the binding screws for adjusting work in jig.

F — Table.

G — Hole in piece *B* that is being drilled in the duplicate.

H — Drilling bushing.

K — Reaming bushing put in position after drilling.

L — Drill $\frac{1}{64}$″ small to allow for reaming.

Digitized by Google

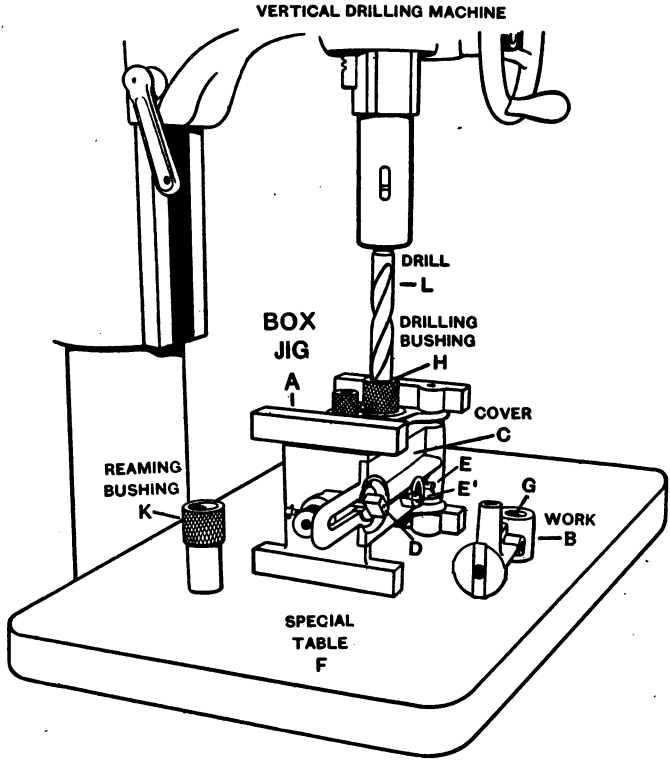


FIG. 334. — A BOX JIG FOR DRILLING, REAMING, AND TAPPING IN DIFFERENT DIRECTIONS.

RADIAL DRILLING MACHINES.

505. Radial drilling machines, commonly called radial drills, differ from vertical drilling machines in that the drill is moved to aline with the work, which is more convenient for large work, such as machine frames, that cannot be moved easily.

A plain radial drilling machine can be used only for vertical drilling, while a universal radial drilling machine may be used not only for vertical drilling but may be adjusted, also, to drill at almost any angle and used with hand or power feed.

Digitized by Google

506. Plain radial drilling machine, Fig. 335.

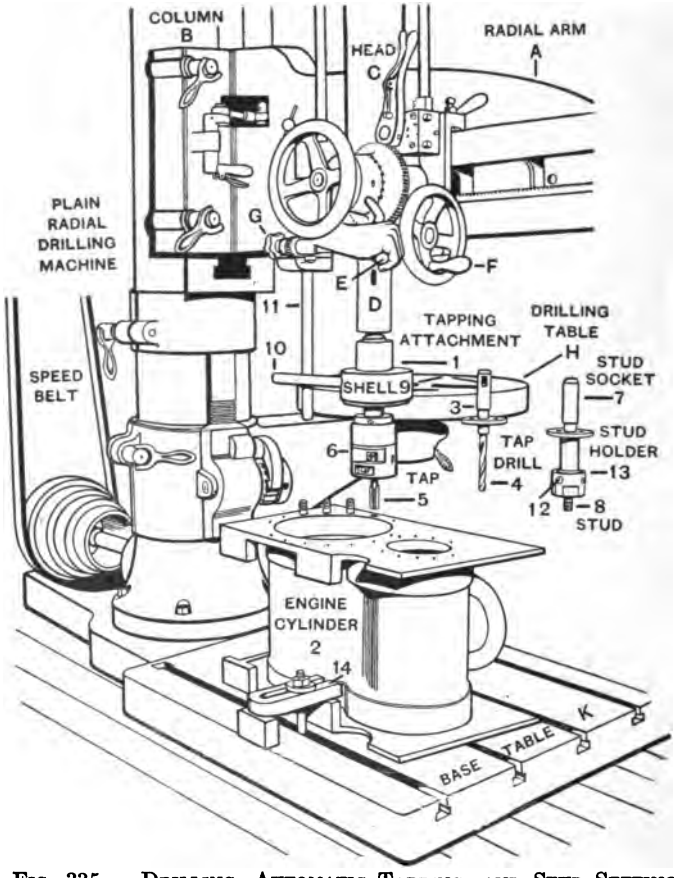


FIG. 335. — DRILLING, AUTOMATIC TAPPING, AND STUD SETTING.

SCHEDULE OF PARTS.

A — Radial arm; may be swung around column by hand, and raised up or down by power.

B — Column.

C — Spindle head; may be moved back and forth on arm *A*.

D — Spindle; may be moved up and down in head *C* by hand.

E — Levers for quick movement of spindle *D*.

F — Hand wheel for slow movement of spindle.

G — Knob operating clutch throwing "in" or "out" power feed.

H — Drilling table for light work.

K — Base table for heavy work.

Digitized by Google

507. Automatic tapping attachment. Friction drive.—1, Fig. 335, is shown fitted to the spindle of a radial drilling machine for drilling, tapping, and setting studs in an engine cylinder 2, without stopping or reversing the drill spindle. Socket 3 holding tap drill 4 is used to drill the hole, and is followed by tap 5 in tap socket 6, which in turn is replaced by stud socket 7 that sets stud 8 in place. Shell 9 holds reversing mechanism and is kept from revolving by rod 10 resting against rod 11. The attachment holds drill sockets and will drill holes in the regular way. To tap, press spindle *D* downward by hand lever *E* until the tap reaches the bottom of the hole as indicated by the slip of the friction drive, then raise handle *E*, which throws in the reversing mechanism and backs out the tap. After setting stud 8, raise handle *E* to back off holder 13, leaving stud 8 in cylinder 2. See *Attention*, § 549.

With some stud holders, stud 8 is released by rapping pin 12 lightly with a hammer before reversing the holder; on others, the release of stud is obtained by a stud nut which operates on the principle that the coarser threads of the holder, when reversed, will release the finer threads of the stud. The cylinder is clamped to base table *K* at 14. For R.P.M. see Power Tapping, § 533.

Attention. — The studs may be set to project to any uniform height by using a gage block between stud holder and cylinder.

Note. — Cap screws, nuts, and slotted screws may be set in like manner by using special wrenches and screw drivers.

508. Jig vise. — In the absence of a regular jig, duplicate work of certain classes may be done with the aid of a jig vise as in Fig. 336. To use this vise lay out and drill one piece and use it as a gage by which to set the stop and jig plate. Vise *A* is heavy and rests without clamping on table *B* of drilling machine *C*. Work *D* is set against adjusting stop *E* and clamped by setting up sliding jaw *F* with lever *G*. Jig plate *H* carrying removable bushing *K* is then adjusted to the desired position for the hole and clamped to fixed jaw of vise. A radial drilling machine being used, spindle *L* and

Digitized by Google

drill *M* may be moved to suit the position of work. Cylindrical distance gage *N*, cut to its center, is sometimes used to set the center of the hole in the jig a given distance from the fixed jaw.

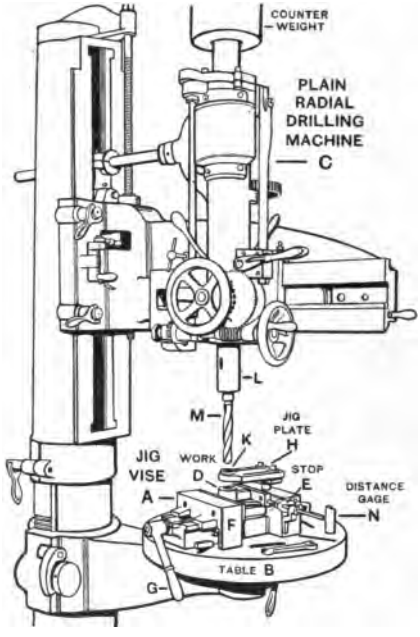


FIG. 336. — DRILLING WITH JIG VISE.

509. To drill and counterbore duplicate parts. — In Fig. 337, a multiple-spindle drilling machine is arranged to drill and spot face casting, as *A*, *A'* held by jig *B*. Insert drill *C* in spindle *D* and drill a hole until arrested by stop *E*. A smaller drill in spindle *F* finishes the hole as shown in section at *F'*. The jig is turned over and drill *G* used to drill the holes in lugs *H* and *H'* through bushings *K* and *K'*. Counterbore *L* is fitted to spindle *M* for spot facing lugs *H* and *H'* for the screw or bolt heads. Fixture *N* of the jig is used to clamp the rod of the other half of the strap while it is being drilled.

Digitized by Google

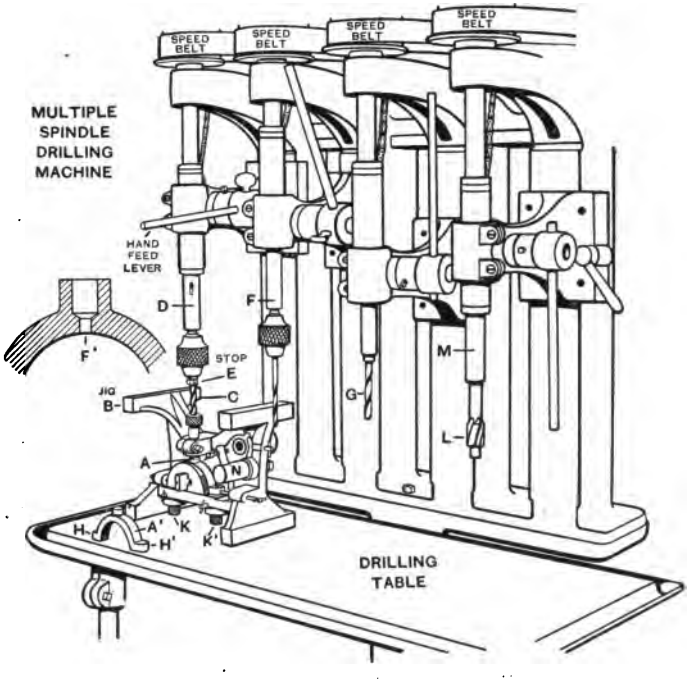


FIG. 337. — DRILLING AND COUNTERBORING DUPLICATE PARTS.

TAPS AND DIES.

510. A tap cuts inside threads. Taps are obtainable in all standard sizes, right or left, either United States Standard or Sharp V threads, and are designed for use by hand or power. 29°, Ratchet, and Square threads are cut with special taps.

511. Sizes of taps. — Taps are obtainable from $\frac{1}{4}''$ diameter to 3″. See table of U. S. S. and Sharp V-Thread Screws, Taps, etc., § 538. Small taps and screws less than $\frac{1}{4}''$ in diameter are obtainable from $\frac{1}{16}''$ to $\frac{9}{32}''$ by 64ths in Sharp V threads only. See § 574, and Machine Screws, §§ 569–573.

512. Set of hand taps, Fig. 338. — Taper tap A is turned off at end *D* to root of thread, to enter the drilled hole and also

Digitized by Google

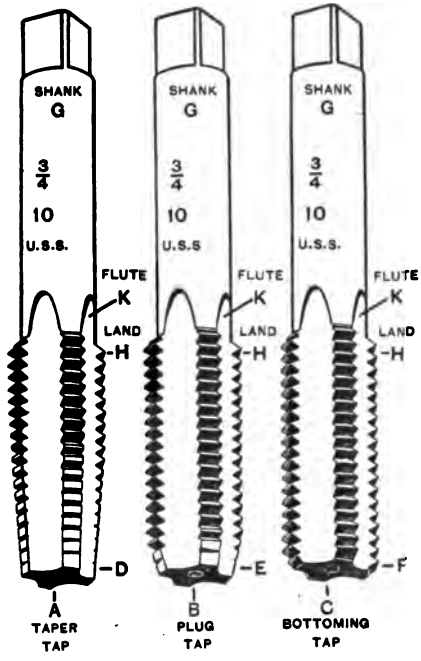


FIG. 338. — SET OF HAND TAPS.

to cut the thread gradually. For tap drills, see §§ 538, 569, 570, 573, 574. Plug tap *B* is tapered a little at end *E*. Bottoming tap *C* is not tapered at *F*.

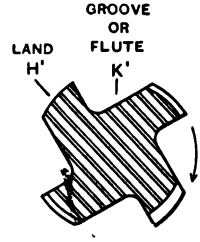


FIG. 339. — SECTION OF RIGHT TAP.

Shank *G* of all ordinary taps is made the same diameter or smaller than root of thread. For "jig" tapping, the shank is made to full diameter of thread. Threaded parts *H* are the "lands" or teeth. Flutes or grooves *K* form a cutting edge for face of the thread and an outlet for chips. The diameter and threads per inch and name of thread are stamped on each, as "$\frac{3}{4} \times 10$, U. S. S." Fig. 339 is a section of a right tap, the land at *H'* and the flute at *K'*.

Digitized by Google

513. Clearance on taps. — The first two or three threads are given clearance or "backed off." On large taps the sides of the teeth are "backed off" by special machines, or by hand with a file.

514. Nut taps are made with long shanks and long threaded parts, and in some cases the thread is cut taper on the end for about half the length of the threaded portion, so that cutting will be gradual.

FIG. 341. — ADJUSTABLE TAP.

515. Pulley taps are long so that the wrench may rotate when tapping hub of pulley, whether tap passes through rim of pulley or by its edge.

516. Hob or master taps, Fig. 340, are for tapping solid bolt dies, adjustable dies, screw plates, etc. Taper $\frac{3}{16}''$ to $\frac{1}{4}''$ per foot.

517. An adjustable hand tap, Fig. 341, is split and adjusted by taper screws *A* and *A'*, and the two parts locked by screws *B* and *B'*. Such a tap is used to enlarge threaded holes.

518. Automatic taps are used in turret lathes, screw machines, etc., by power. When desired length of thread is cut the tap is closed by hand or automatically, and withdrawn, avoiding reversal of work. See *Elements of Machine Work* and *Advanced Machine Work*.

FIG. 340. — HOB OR MASTER TAP.

519. Taps are sharpened by grinding the front of the teeth radial with a grinding wheel whose edge has been rounded over with a diamond tool. They may be hand or machine ground.

520. Tap (reamer) wrenches. See Fig. 344. — In the adjustable tap wrench the jaws are tightened on the tap, so that

Digitized by Google

there will be no loosening of the grip during the operation of tapping.

521. Adjustable Tee tap wrench for small taps, Fig. 342. — *A* carries chuck *B* adjusted on tap *C*, and is shown tapping nut *D* held in vise *E*.

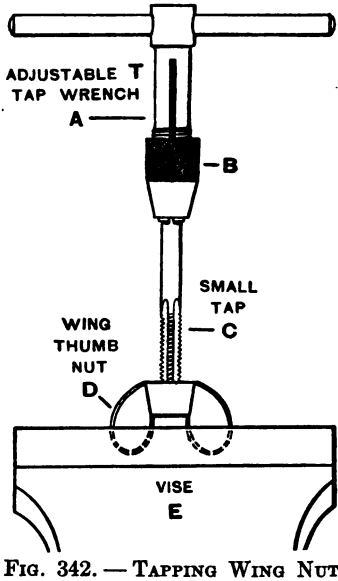


FIG. 342. — TAPPING WING NUT.

522. Single-end tap wrench, Fig. 343. — To tap a hole in a corner the tap is steadied with the left hand and rotated with the right by means of a single-end tap wrench or a ratchet wrench. A ratchet wrench is usually operated with one hand, thus lessening the liability of breaking the tap.

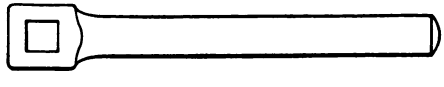


FIG. 343. — SINGLE-END TAP WRENCH.

523. Tap holes are either drilled in solid stock, punched, cored and drilled, or cored and bored. As a screw or bolt should enter a tapped hole in the frame of a machine about $1\frac{1}{2}$ times its diameter, the hole is drilled to a depth of twice the diameter of the screw and then tapped to a depth of about $1\frac{3}{4}$ times its diameter.

Digitized by Google

Attention. — Do not start a tap in a cored hole, as the surface is hard, and covered with sand and scale. This method would dull the tap.

524. Loose nuts. See Alinement Tapping, §§ 626, 627. Less than 1″ are threaded in a vise with a tap. Large nuts are threaded in a lathe, or rough threaded in a lathe and finished with a tap. See § 594.

Nuts that are to be finished are tapped as accurately as can be determined by the eye. Then they are squared at a right angle to the axis of the thread. See Nut Mandrels, § 256.

525. Hand tapping a rough nut, Fig. 344. — The holes in machine steel or wrought iron, hexagonal and square cold-pressed nut blanks, usually come punched the correct diameter for tapping.

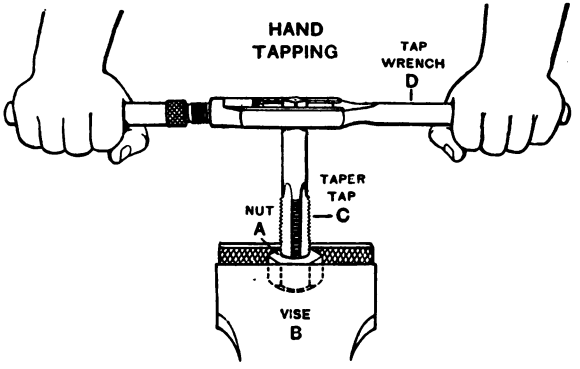


FIG. 344. — TAPPING HEXAGONAL NUT BLANK. APPROXIMATE TAPPING.

SCHEDULE OF OPERATIONS.

1. Place nut *A* level in vise *B* and drop lard oil on threads of right tap *C*.

2. Place wrench *D* on tap and hold tap perpendicular to face of nut. Rotate wrench *D* to right, at same time pressing hard, downward, until thread catches or it may ream hole.

3. Sight tap in front and at side; if it is out of true, correct by turning tap backward, then forward, exerting correcting side pressure on forward stroke only.

4. Oil tap freely. After each forward stroke, back tap part of a turn to allow chips to drop out and oil to reach teeth.

Digitized by Google

Attention. — Press evenly on both handles and make tap cut true while first few threads are being cut. To make correction after threads are firmly caught may break the tap.

A tap that is difficult to start may be started dry, then oiled. Small taps, $\frac{3}{4}''$ and less diameter, spring considerably before they break. When a tap begins to spring, back it out, clean, and oil.

526. Accurate hand tapping. Three methods. Figs. 345, 346, 347.—

Holes in pillow blocks, frames of machines, etc., must be tapped in alinement to the machined surface or the bolts, cap screws, stud bolts, etc., will bind and may prevent them from being bolted in place. The most accurate tapping is done in a drilling machine before work is unclamped, Fig. 345.

If more expedient to remove work from drill press before tapping, a tapping jig may be used as shown in Fig. 346. In the absence of a tapping jig, a hole may be accurately tapped by testing with a square, as shown in Fig. 347.

SCHEDULES OF OPERATIONS, MACHINES AND TOOLS.

527. Tapping in vertical drilling machine, Fig. 345.

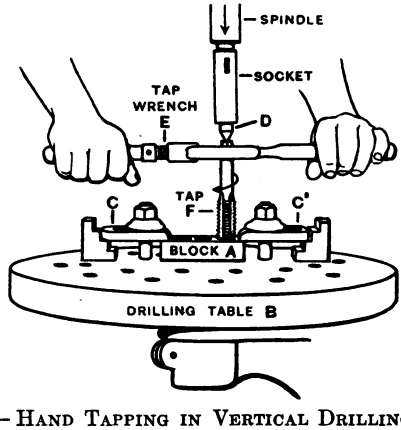


FIG. 345. — HAND TAPPING IN VERTICAL DRILLING MACHINE. ACCURATE TAPPING.

Digitized by Google

1. Clamp block *A* to table *B* by clamps *C* and *C'*. Clamp table. Drill hole to desired diameter.

2. Remove drill and insert center *D* on spindle. Place wrench *E* on tap *F*, insert in hole and bring center *D* down into countersink in end of tap.

3. Tap in regular way, keeping tap central by following it with center *D*.

528. Jig tapping, Fig. 346.

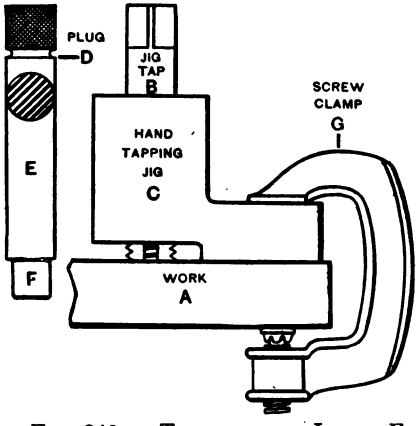


FIG. 346. — TAPPING WITH JIG. ACCURATE TAPPING.

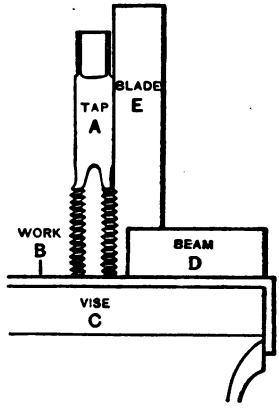


FIG. 347. — TESTING TAP WITH TRY SQUARE. ACCURATE TAPPING.

1. Aline work *A* and jig *C* by plug *D* having a diameter at *E* the same as that of the tap and at *F* the same as that of the hole in the work.

2. Fasten *A* and *C* together with clamp *G*, remove plug and tap in regular way with jig tap *B*.

Attention. — The hole in jig is the full diameter of tap.

529. Testing tap with try square, Fig. 347.

1. Insert right taper tap *A* in hole in work *B* held in vise *C*; rotate to right and press downward hard until thread catches.

2. Remove wrench and test alinement of tap with square by placing beam *D* on work and blade *E* against shank of tap. The nature of light line between blade and shank of tap will indicate correction to be made.

Test with square at two positions at 90° to each other.

3. If tap is not in alinement, correct as in § 525.

Attention. — Test with square frequenlty during operation of starting tap.

Digitized by Google

530. Tapping "through" holes and "bottom" holes. — To tap a "through" hole, as a nut, use a taper tap. To tap a "bottom" hole, use three taps: first, the taper tap; second, the plug tap; third, the bottoming tap. When the nature of the work will admit, drill the hole deeper than the length of screw to avoid using a bottoming tap.

531. To make a tap cut large. — In the absence of a large tap, or an adjustable tap, roll cotton waste as at *A*, Fig. 348, and insert it into grooves of tap *B*, then run the tap through nut *C* held in vise *D*. If necessary, repeat with more waste or a narrow strip of sheet copper or brass.

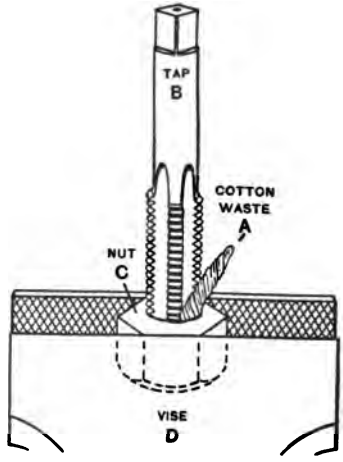


FIG. 348. — MAKING A TAP CUT LARGE.

532. Removing broken taps. — The skillful use of a center punch or chisel and hammer will sometimes remove a broken tap. If the work is small, it may be annealed and drilled out.

Fig. 349 shows a mechanical method of removing a broken tap. Box *A* held in vise *B* shows a broken tap *C*. Four fingers of tap remover *D*, two of which are shown at *E* and *E'*, are pushed down into grooves of tap by collar *F*. Sleeve *G* is then pressed down close to work and tap removed by rotating tap wrench *H*.

Digitized by Google

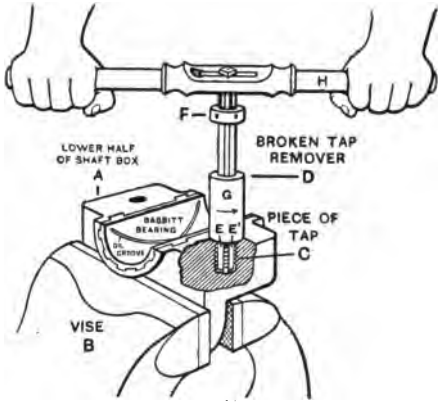


FIG. 349. — REMOVING A BROKEN TAP.

533. Power tapping. — Taps are used by power in drilling machines, boring machines, turret lathes, and nut-tapping machines. See §§ 506, 507, also *Advanced Machine Work*. Taps may be used safely at from one-third to one-half the R.P.M. of drills (see §§ 454–457) for holes not deeper than $1\frac{1}{2}$ times diameter of tap. For deeper holes the speed must be less.

584. Tapping and threading sizes. — To avoid breaking the tap or tapping a full thread, which is not desirable, especially on cast iron, the tap drill or bore of hole in lathe work should be *larger* than the theoretical root diameter of tap. To avoid a full thread on a screw threaded in an engine lathe or a full and torn thread when cut with a die, the screw blank should be *smaller* than the nominal diameter of screw.

The amount that the tap drill should be large or screw blank small varies for different classes of work and with different users, but in general it is for United States Standard thread from .004″ for a $\frac{1}{4}$″ thread to .010″ for a 3″ thread; and for a Sharp V thread, from .010″ for a $\frac{1}{4}$″ thread to .060″ for a 3″ thread.

Attention. — When United States Standard thread taps, which are used to tap the nuts, are made to a larger diameter at the top of the thread to allow for clearance, it is unnecessary to turn the screw blank small.

Digitized by Google

535. Diameter of a tap drill, *A*, Fig. 350, equals the diameter at root of thread, as at *B*, for United States Standard and Sharp V-thread taps plus the allowances given in § 534. The approximate method of finding this is to measure the root diameter of the tap with thin, pointed calipers (see § 216), but the more accurate method is by calculation.

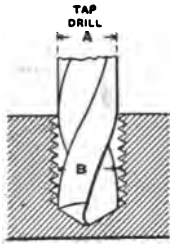


FIG. 350. — RELATION OF DRILL TO TAPPED HOLE.

536. To calculate root diameter for Sharp V thread.

Constant: 1.732 = double depth of a 1″ P Sharp V thread.

Formula:

$$\text{Outside diam. of tap} - \frac{1.732}{\text{No. threads per inch}} = \text{diam. of tap drill.}$$

Example. — Find diameter of drill for $\frac{3}{4}$″ tap, 10 Sharp V threads per inch.

Solution. — $.750 - \frac{1.732}{10} = .5768.$

Use $\frac{39}{64}$″ tap drill. See § 33 and Table, § 538.

537. To calculate root diameter for United States Standard thread.

Constant: 1.299 = double depth of a 1″ *P* United States Standard thread.

Formula:

$$\text{Outside diam. of tap} - \frac{1.299}{\text{No. threads per inch}} = \text{diam. of drill.}$$

Example. — Find diameter of drill for $\frac{1}{4}$″ tap, 20 United States Standard threads per inch.

Solution. — $.250 - \frac{1.299}{20} = .185$, diameter of drill.

Use $\frac{3}{16}$″ tap drill.

Digitized by Google

538. Table of United States Standard and Sharp V-thread screws, taps, and tap drill sizes which are also the sizes to bore holes for inside threads.

Diameter of Tap.	No. of Threads per Inch.		Sizes of Tap Drills.	
	U. S. S.	Sharp V.	U. S. S.	Sharp V.
$\frac{1}{4}$	20	20	$\frac{3}{16}$ or 11	$\frac{3}{16}$ or 14
$\frac{5}{16}$	18	18	$\frac{15}{64}$ or C	$\frac{15}{64}$ or 1
$\frac{3}{8}$	16	16	$\frac{19}{64}$ or N	$\frac{19}{64}$ or L
$\frac{7}{16}$	14	14	$\frac{23}{64}$ or S	$\frac{23}{64}$ or R
$\frac{1}{2}$	.....	12	.....	$\frac{25}{64}$
$\frac{1}{2}$	13	13	$\frac{13}{32}$	$\frac{25}{64}$ or X
$\frac{1}{2}$	.....	14	.....	$\frac{13}{32}$
$\frac{9}{16}$	12	12	$\frac{29}{64}$	$\frac{29}{64}$
$\frac{5}{8}$	11	11	$\frac{33}{64}$	$\frac{1}{2}$
$\frac{11}{16}$	11	11	$\frac{37}{64}$	$\frac{9}{16}$
$\frac{3}{4}$	10	10	$\frac{5}{8}$	$\frac{39}{64}$
$\frac{13}{16}$	10	10	$\frac{11}{16}$	$\frac{43}{64}$
$\frac{7}{8}$	9	9	$\frac{47}{64}$	$\frac{23}{32}$
$\frac{15}{16}$	9	9	$\frac{51}{64}$	$\frac{25}{32}$
1	8	8	$\frac{27}{32}$	$\frac{53}{64}$
$1\frac{1}{8}$	7	7	$\frac{61}{64}$	$\frac{59}{64}$
$1\frac{1}{4}$	7	7	$1\frac{3}{64}$	$1\frac{3}{64}$
$1\frac{3}{8}$	6	6	$1\frac{11}{64}$	$1\frac{1}{8}$
$1\frac{1}{2}$	6	6	$1\frac{19}{64}$	$1\frac{17}{64}$
$1\frac{5}{8}$	$5\frac{1}{2}$	5	$1\frac{25}{64}$	$1\frac{21}{64}$
$1\frac{3}{4}$	5	5	$1\frac{1}{2}$	$1\frac{29}{64}$
$1\frac{7}{8}$	5	$4\frac{1}{2}$	$1\frac{5}{8}$	$1\frac{35}{64}$
2	$4\frac{1}{2}$	$4\frac{1}{2}$	$1\frac{23}{32}$	$1\frac{43}{64}$
$2\frac{1}{4}$	$4\frac{1}{2}$	$4\frac{1}{2}$	$1\frac{31}{32}$	$1\frac{59}{64}$
$2\frac{1}{2}$	4	4	$1\frac{3}{16}$	$2\frac{1}{8}$
$2\frac{3}{4}$	4	4	$2\frac{7}{16}$	$2\frac{3}{8}$
3	$3\frac{1}{2}$	$3\frac{1}{2}$	$2\frac{11}{64}$	$2\frac{9}{16}$

Attention. — For screws and taps smaller than $\frac{1}{4}''$, see §§ 565–574.

539. A die cuts outside threads on screws, bolts, rods, etc., usually with one cut by hand or power, and this size is designated by the diameter of the screw it will cut. Right or left, United States Standard, Sharp V, 29°, and Ratchet threads

Digitized by Google

can be cut with dies, but Square threads cannot be successfully cut by them.

Dies are made by drilling and tapping die blank with taper hob or master tap (see § 516) from the back to give it body clearance.

The flutes are made by drilling and filing, or milling to produce radial cutting edges for general work. Negative cutting edges are used for brass. The muzzle or cutter side is tapered or chamfered to enable die to start centrally and cut thread gradually.

540. Sizes of dies. — Solid and separate dies are obtainable in all standard sizes in United States Standard or Sharp V threads to match taps. See Table, § 538.

541. Automatic dies are used in turret lathes, screw machines, and bolt cutters operating by power. They are made in sections and held in die head so that when desired length of thread is cut die is opened by hand, or automatically, and withdrawn, avoiding reversal of work.

542. Solid dies are sometimes sharpened by grinding with a small grinding wheel. See *Advanced Machine Work.*

Dies made in halves are sharpened with a thin wheel, and chasers, of which automatic dies are composed, are ground separately with an ordinary wheel.

543. Power (die) threading. — Dies are used by power in drilling machines, screw machines, turret lathes, and bolt cutters. They are run at a cutting speed of one-third that of drills, see §§ 454–457, or from 10 to 20 F.P.M. This speed may be increased if material is very soft, as screw machine stock, and abundant lubricant is used.

Attention. — Carelessly threading with a die or without a lubricant at too high a speed, as 75 or 100 F.P.M., would injure a die so badly that it could not be used at a higher speed than 6 or 8 F.P.M. until sharpened or replaced by a new one. Slow speed and an abundant supply of lubricant keep the die cool, which is necessary for accurate work and the preservation of the die.

Digitized by Google

544. Hand (die) threading. — To thread bolt by hand with die and die stock. Fig. 351.

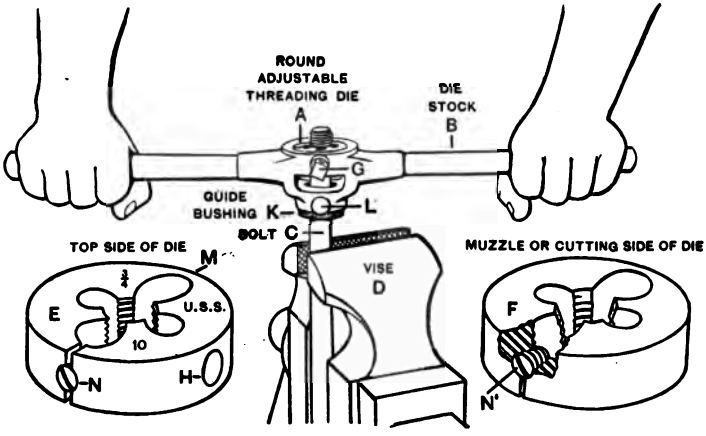


FIG. 351. — THREADING BOLT OR ROD BY HAND.

SCHEDULE OF OPERATIONS.

Place die *A* in stock *B* and hold bolt *C* in vise *D*. Die is shown in detail at *E*, *F*. Top side *E* has full thread to surface and is stamped "$\frac{3}{4}'' \times 10$, U. S. S." Place die muzzle side down in stock and gently fasten by set screw *G*, entering cavity *H*. Put guide bushing *K* in bottom of stock and fasten by thumb screw *L*. File an even bevel on end of bolt to start die. Oil end of bolt through top of die and force die onto bolt with downward pressure and rotation of handles to the right until die cuts a few threads, after which it will not need downward pressure but will feed itself. Oil, and occasionally rotate backward to allow chips to drop out and oil to reach cutting edges. To terminate thread abruptly, turn die and stock over and use side *E*. Die is spring-tempered at *M*, and adjusted by taper screw at *N*, *N'*. Some dies have taper pin adjustment and in place of bushing have adjustable jaws to guide die.

Attention. — Dies are often used to size screws that have been rough threaded in the lathe, as they finish work nicely, produce uniform sizes, and save time.

Digitized by Google

CHAPTER XVII.

BOLTS AND NUTS. CAP AND SET SCREWS. MACHINE SCREWS. COUNTERBORING AND COUNTERSINKING. CALCULATING DIAMETER OF BLANK TO MILL SQUARE OR HEXAGONAL. INDEXING IN ENGINE LATHE.

BOLTS AND NUTS.

545. A bolt, Fig. 352, is a rod of metal with a head at one end and a thread and nut at the other. It is used to fasten together two or more pieces of any material. Bolts are

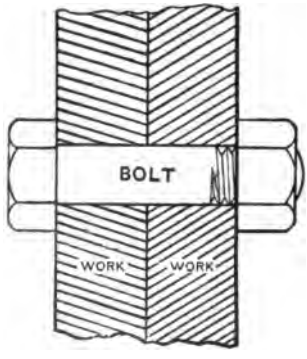


FIG. 352. — CLAMPING WITH BOLT.

made of steel, iron, and bronze. See Table of United States Standard Bolt Heads and Nuts, §§ 641, 642.

The length of the bolt is measured from under the head to the point, therefore a bolt will clamp its length minus the thickness of the nut. Bolts under 20 inches are usually threaded about 2 times their diameter; over 20 inches, about 3 times.

While the most common forms of heads are square or hexagonal, as in Fig. 353, bolts with various styles of heads are obtainable, as eye bolts, bolts with skew heads, eccen-

Digitized by Google

tric heads, key heads, etc. Bolts may be obtained with brass, copper and bronze heads electrically welded to steel bodies, and are largely used in electrical and government work. See Cap Screws § 557.

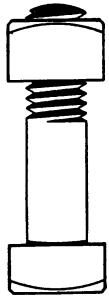
There are three classes of bolts:

First, rough bolts which are forged and threaded without machining.

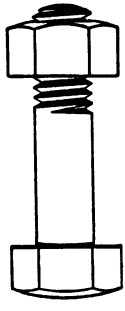
Second, semi-finished bolts with body and under-head and nut machined.

Third, finished bolts which are machined all over.

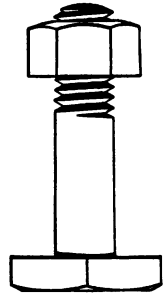
They are obtainable rough forged and machined, or machined from bar stock.



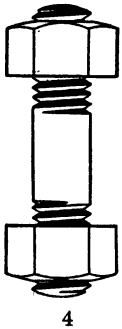
1
SQUARE HEAD.



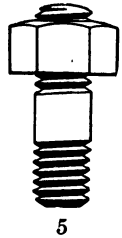
2
HEXAGONAL HEAD.



3
PLANER HEAD.



4
IMPROVISED BOLT.



5
STUD BOLT.

FIG. 353. — DIFFERENT KINDS OF BOLTS.

Digitized by Google

546. Planer bolts, Fig. 353–3, have thin and flat hexagonal, or square heads to fit Tee slots, and are used to bolt work to planer and other machine tables. A rough iron washer is generally used under the nut. Thin nuts are obtainable.

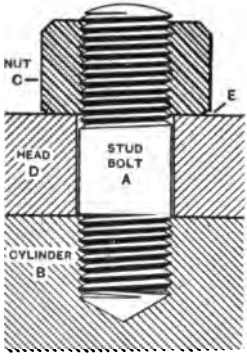


FIG. 354. — USING STUD BOLT AS A FASTENING.

547. An improvised bolt, Fig. 353–4, often called a double-end bolt, may be made quickly by threading both ends of a rod and using two nuts.

548. Stud bolts, Fig. 354, have one end threaded to screw hard into work *B* and the other to receive nut *C* to fasten head *D* to cylinder. As the corners of nuts and bolt heads scar finished work, the sides are often squared down to the flats as at *E*.

549. Setting studs. — Studs may be set in place by hand or power. See §§ 506, 507. To set studs by hand use a stud holder, Fig. 355, which consists of block *A*, screw *B*, and lock nut *C*. Stud *D* is screwed into the cylinder head by hand. Holder *A* is screwed on stud *D* any desired distance, and screw *B* is set against stud *D* and locked with nut *C*; then a wrench is used on the holder to set the stud. The height of the studs in the cylinder head should be uniform and is determined by the distance stud *D* is screwed in holder *A*, and height of gage block *F*.

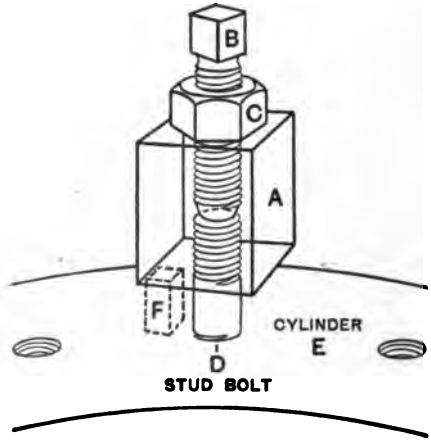


FIG. 355. — SETTING STUD BOLT IN CYLINDER HEAD WITH STUD HOLDER.

Attention. — Always clean and oil a screw before screwing it in nut or work. See § 593.

Digitized by Google

550. Expansion bolt *A*, Fig. 356. — An expansion bolt is used to bolt a machine, plank, or bracket, as at *B*, to brick or stone walls where it is not desirable to drill holes entirely

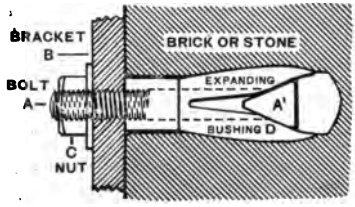


FIG. 356. — USING EXPANSION BOLT AS A FASTENING.

through. As nut *C* is tightened the tapered head *A'* forces expanding bushing *D* against walls of hole. See Lag Screws, § 560.

551. Headless bolts may be secured in holes in a stone foundation by scarring, running in hot lead, and calking. Sulphur or type metal is sometimes used and not calked.

552. A nut, Fig. 357, is a block of metal which may be of any shape with a threaded hole to receive the threaded end of a bolt, rod, or screw. While the most common forms of nuts are square or hexagonal, either cold punched or hot pressed, special nuts of various styles are obtainable, as collar nuts which have to be operated with pin or spanner wrench, and regular and cap or blind nuts combined with the washer for special work, etc.

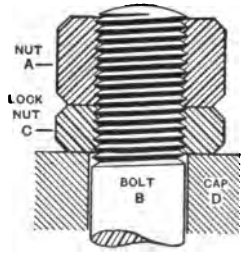


FIG. 357. — LOCK NUT TO PREVENT LOOSENING OF REGULAR NUT.

553. Lock or check nuts, Fig. 357. — When nut as at *A* and bolts as at *B* are subjected to much jarring and in danger of becoming loose, a second nut which may be thinner as at *C*, and called a "lock" or "check" nut, is first put on and screwed to bind hard on the work or cap *D*, and held with

Digitized by Google

a wrench while the regular nut *A* is screwed down hard against it. If a loose connection is desired on the cap or other work, the regular nut *A* is held with a wrench and lock nut *C* backed hard against it.

554. Spring cotter or split pin. — A method used to prevent the nut from becoming lost is to insert a spring cotter into a hole bored through the end of the stud or bolt above or in back of the nut. The ends of the cotter are spread apart, which prevents it from dropping out.

555. A castle (automobile) nut, Fig. 358, is a still safer method. Bolt *A* protrudes through the nut *B* which is provided with diametrical slots. Spring cotter *C* passes through a slot and through a hole in the bolt, and its points are spread as at *D*. Spring keys similar to the split pin *C*, made of flat steel, are much used. See §§ 639, 640.

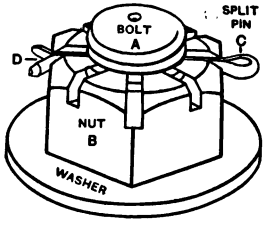


FIG. 358. — CASTLE NUT TO PREVENT UNFASTENING.

A spring cotter alone prevents an ordinary nut from backing off, and if used with a castle nut prevents the nut from moving either way.

556. Turnbuckles are used to connect and regulate the length or tension of truss rods, tie rods, pipes, or wires. They are either machined from hexagonal stock as at *A* in Fig.

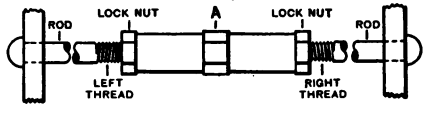


FIG. 359. — A TURNBUCKLE TO CONNECT AND REGULATE TENSION OF TRUSS RODS AND TIE RODS.

359, or forged and turned and supplied with right and left threads or with a right thread on one end and a swivel on the other. Lock nuts are sometimes used as shown.

Digitized by Google

CAP AND SET SCREWS.

557. Cap screws are used to fasten one piece of metal not threaded to another that is threaded, as in Fig. 360. They

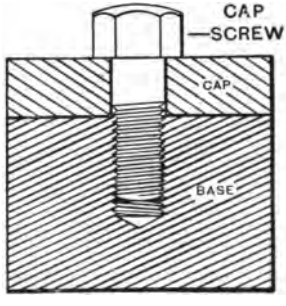


FIG. 360. — CLAMPING WITH CAP SCREW.

differ from a bolt in that they are not supplied with a nut. See § 473. They are made in steel, iron, or brass, with differently shaped heads such as square, hexagonal, round, flat, etc., Fig. 361. Cap screws with brass, copper, and bronze heads electrically welded to steel bodies are also obtainable. See Bolts, § 545.

Cap screws range in diameters from $\frac{1}{4}''$ to $1\frac{1}{4}''$, United States Standard and Sharp V threads. See Table of Tap Drills, § 538.

The length of square, hexagonal, round, and fillister head cap screws is measured from under head to point. Flat and oval countersunk heads are measured over all. These have an included angle from 72° to 82°. See Machine Screws, § 565.

There is no universal standard of heads, but different screw manufacturers have standards of their own which differ from one another. See Counterboring, § 575, and Countersinking, § 579.

To avoid special wrenches, cap screws with heads the same size as United States Standard bolt heads and nuts are obtainable. See §§ 640, 641.

Digitized by Google

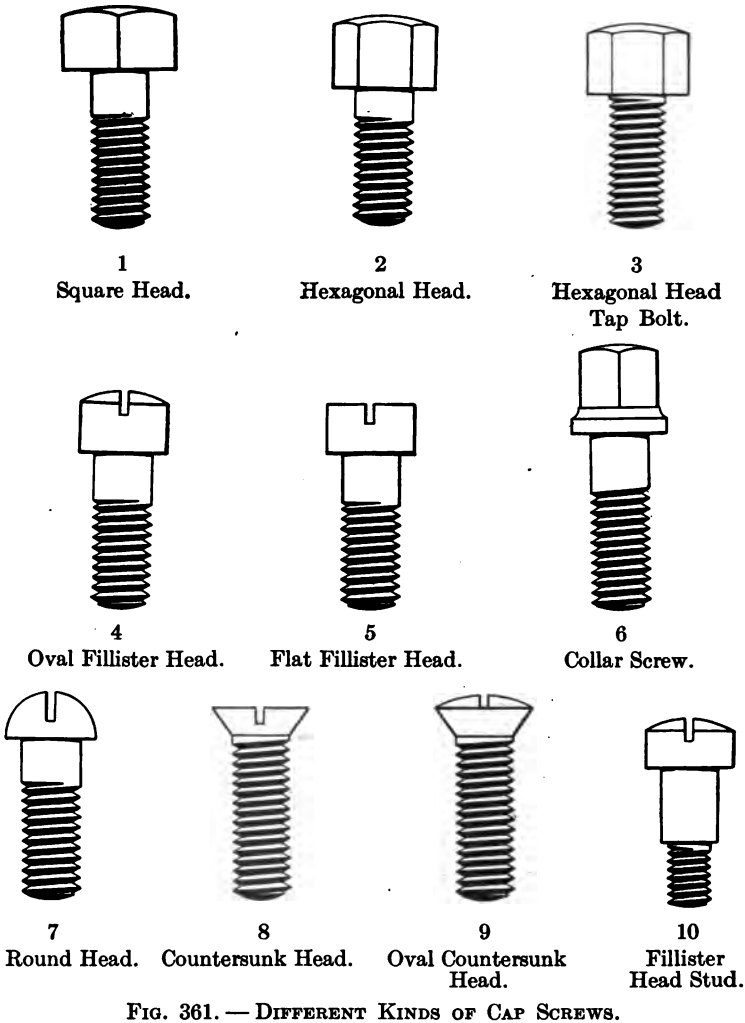


FIG. 361. — DIFFERENT KINDS OF CAP SCREWS.

Short screws are usually threaded to the head. Long screws are threaded about two-thirds the length of the body.

558. Tap bolts, Fig. 361, are cap screws threaded to the head to clamp a thin piece of work to a thick one.

Digitized by Google

559. Dowel pins to insure alinement in erecting machinery. — These pins are especially useful where machines have to be taken apart and re-erected, as they preserve alinement independent of the cap screws or bolts.

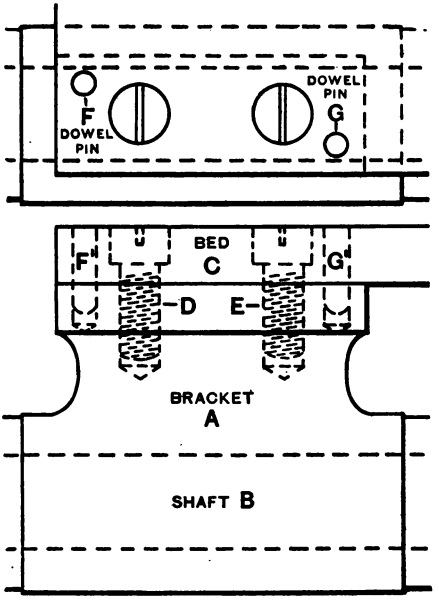


FIG. 362. — USE OF DOWEL PINS.

Bracket *A*, Fig. 362, forming a bearing for shaft *B*, is fastened to bed *C* by screws *D* and *E*. To always insure alinement of the bearing after once adjusted, use at least two pins as at *F*, *F'* and *G*, *G'*. They are usually made a driving fit in one part and an easy fit in the other part. On some classes of large work taper dowel pins are used to fit taper holes.

560. Lag screws, *A*, Fig. 363, are used to fasten machines to the floor or light hangers, as *B* to hanger plank *C*. Bore holes for lag screws slightly larger than the root diameter of the screw. Lubricate the screw with soap, tallow, or oil,

Digitized by Google

start by driving slightly. Split tapering sleeves of malleable iron with projections to grip the walls of the hole, and with inside threads to fit lag screws, are obtainable, and are used as expansion bolts. See Expansion Bolts, § 550.

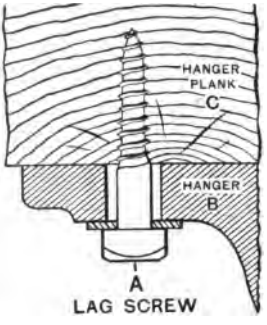


FIG. 363. — FASTENING A HANGER WITH A LAG SCREW.

561. Hanger bolts, for fastening large hangers and pillow blocks, are threaded as a lag screw at one end and as a bolt at the other to receive a nut, and are operated with a pipe wrench, or by locking two nuts together on the threaded portions and using a monkey wrench.

562. Wood screws are obtainable in diameters from Nos. 0 to 30 by the American Screw Company's gage, Fig. 372, and in length from $\frac{1}{4}''$ to 6″. The length, except round heads, is measured over all, and the threaded portion is about three-fifths of the length. Round heads are measured under the head but come a little short. For example, 1″ and under will come about $\frac{1}{16}''$ short, and over an inch about $\frac{1}{8}''$ short. There is no standard number of threads per inch. Flat and oval countersunk heads have an included angle from 72° to 82°. See Countersink, § 579. These screws are designated thus:

$1\frac{1}{2}''$ × 10, flat head, bright.
$1\frac{1}{2}''$ × 9, round " , blued.
1″ × 6, flat " , brass, etc.

Digitized by Google

Wood screws are often used to fasten metal to a wooden backing, Fig. 364. Flat-head screw *A* fastens wrought-iron bar *B* to wood *C*. A body hole is drilled through metal *B* and countersunk. As a wood screw makes its own nut, it is often permissible to force a small screw into soft wood without boring a hole; still, for a large screw, or hard wood, or nice work, a hole should be bored to nearly the size of the root diameter of the screw, and the screw lubricated. For rough work and soft wood a regular screw is often driven from one-half to two-thirds its length with a hammer, then turned to its seat with a screw driver. A special drive wood screw is obtainable which may be driven its entire length with a hammer. It is removed with a screw driver.

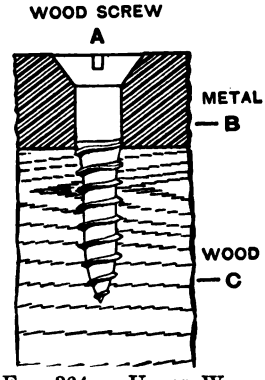
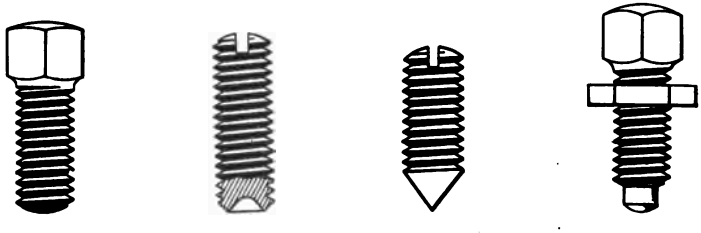


FIG. 364. — USING WOOD SCREW AS A FASTENING.

Wood screws are supplied with lead nuts and are often used as expansion bolts in marble or slate. See Expansion Bolt, § 550.

To fasten metal fixtures to slate or marble, wood screws and litharge are also used. A body hole is drilled in the slate or marble, which is filled with litharge mixed with boiled linseed oil or glycerine and the screw is forced into this preparation.



1 Round Point. 2 Headless Cup Point. 3 Headless Cone Point. 4 Hanger Pivot Point.

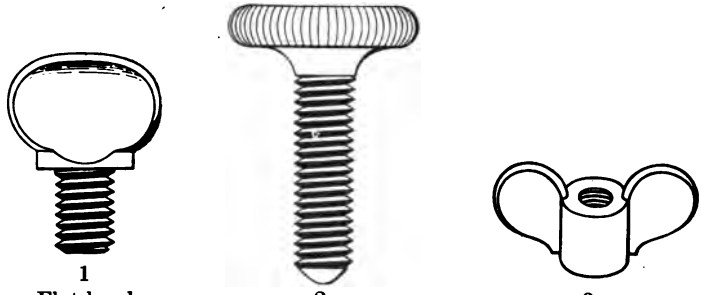
FIG. 365. — SET SCREWS.

563. Set screws, Fig. 365, are used to fasten pulleys to shafts, and for other work of similar nature. See § 317, also

Digitized by Google

Advanced Machine Work. Set screws are obtainable case-hardened, of steel or iron, in diameters from $\frac{1}{4}''$ to $1\frac{1}{4}''$, United States Standard and Sharp V threads. Length of screw is measured from under head to point. Headless screws are measured over all.

564. Thumb screws and wing nut, Fig. 366. — Thumb screws and wing nuts are obtainable in steel, iron, and brass,



1 Flat-head Thumb Screw. 2 Nurled Head. 3 Wing Thumb Nut.

FIG. 366. — THUMB SCREWS AND WING NUT.

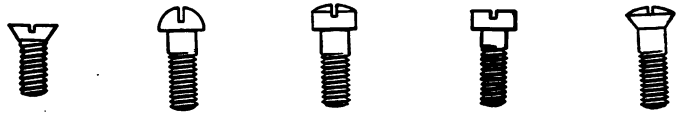
and range in diameter from $\frac{1}{8}''$ to $1''$ in regular sizes and from 0 to 30 in machine screw sizes. See § 565. Length of screw is measured from under the head to the point.

MACHINE SCREWS.

565. Numbered screws, called machine screws, and designated thus: 10×30, meaning 10 size and 30 threads per inch, are used to obtain a greater variety of sizes in screws less than one-half inch in diameter instead of those measured in binary fractions as $\frac{3}{16}'' \times 30$. See § 574. There are two standards: The American Society of Mechanical Engineers' Standard Machine Screw (see § 569), which has the United States Standard form of thread, and the American Screw Company's Machine Screw (see § 573), which has the Sharp V form of thread. The numbers range from 0 to 30, with *standard* and *special* number of threads per inch. See Gage, Fig. 372.

Digitized by Google

They are used in general manufacturing, experimental work, and in building apparatus. The screws are obtain-



1	2	3	4	5
Countersunk Head.	Round Head.	Oval Fillister Head.	Flat Fillister Head.	Oval Countersunk Head.

FIG. 367. — MACHINE SCREWS.

able in steel and iron either bright or blued, and in brass, as in Fig. 367.

The sizes of the screw heads of both standards are nearly the same. The length of the screw is measured the same as are cap screws, and the length of the threaded portions and the angle of the countersink heads are also the same. See Cap Screws, § 557. See Wood Screws, § 562. For Counterboring, see § 575. For Countersinking, see § 579. To use a machine screw as a bolt a hexagonal or square nut is obtainable, and applied as in Fig. 368.

FIG. 368. — MACHINE SCREW BOLT.

566. Machine screw taps are obtainable in standard and special numbered sizes the same as machine

FIG. 369. — MACHINE SCREW PLUG TAP.

screws. A plug tap, as in Fig. 369, is the most used. Taper or bottoming taps are seldom needed. The taps may be used to tap work held in a chuck in a speed lathe, § 338, or the tapping may be started in the lathe and finished at the

Digitized by Google

vise. Fig. 370 shows method of tapping a hole in a collar for a machine screw.

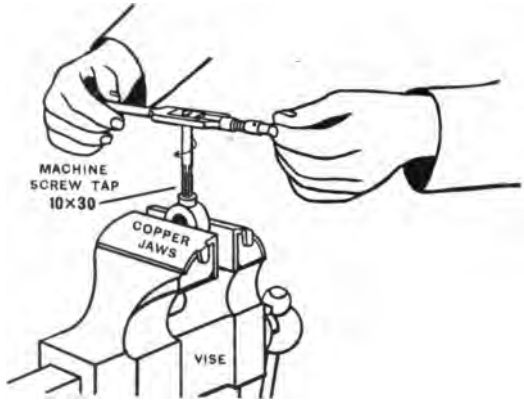


FIG. 370. — TAPPING A COLLAR WITH A 10 × 30 MACHINE-SCREW TAP.

Warning. — As the smaller sizes of these taps break very easily, use a tap drill large enough to allow for three-quarters or seven-eighths of a full thread only, and apply an even and delicate pressure on the tap-wrench handle. Rotate tap gently forward and backward and when the muscle sense indicates that the tap is springing, rotate it backward to prevent breaking. See §§ 522–525.

567. Machine screw dies are obtainable in standard and special numbered sizes, the same as machine screws, and usually round and adjustable as at *A*, Fig. 371. By means of the taper pin *B* or a taper screw, the die may be made to cut slightly large or small to obtain a desired fit of screw in nut. The die is held in the die stock by means of a set screw, see § 544.

While regular machine screws are obtainable, it is often desirable to make screws with special heads or of special lengths as at *C* and *C'*, Fig. 371. Rod *D* may be held in a universal chuck *E* (or in one of the spring (draw in) chucks as at *F*) in the hollow spindle universal hand lathe, and turned

Digitized by Google

to desired size and chamfered. Die and die stock G are held against table center H with muzzle or chamfered end of die on screw blank and die stock supported by Tee rest J. The belt K is pulled with left hand to rotate screw blank, and

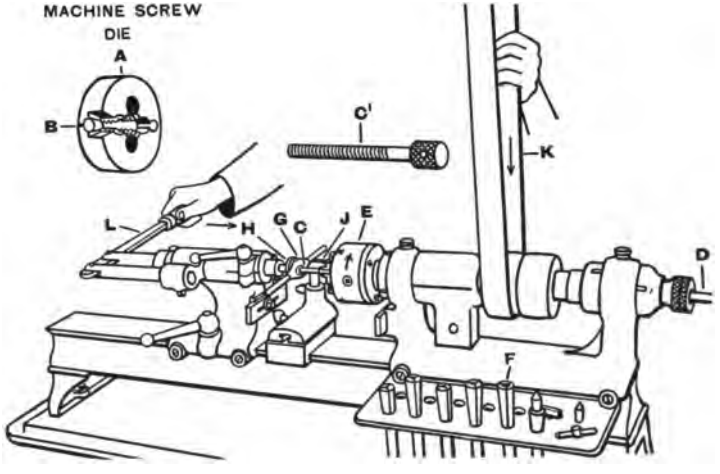


FIG. 371. — THREADING A SCREW WITH 10×30 MACHINE-SCREW DIE IN UNIVERSAL HAND LATHE.

die is forced onto screw blank with lever L operated with right hand.

Attention. — Machine-screw die stocks have no guide, and while a short thread may be cut in a vise with the die without noticeable defect, a long thread would be crooked and not desirable for many purposes.

The thread should be cut in the lathe as in Fig. 371, or started in the lathe and finished in the vise.

568. American Society of Mechanical Engineers' standard machine screws, see §§ 569, 570, have United States Standard form of thread. The difference between consecutive sizes, as 0 to 10, is .013″; and between alternate sizes, as 10 to 30, is .026″.

In order that the screws shall always fit the tapped holes,

Digitized by Google

the taps are slightly larger than the screws, and each is made to maximum and minimum sizes, but the difference between minimum tap and maximum screw is sufficient for error in pitch and wear of tap, insures a good fit and makes the screws interchangeable.

569. Table of American Society of Mechanical Engineers' standard machine screws, taps, and tap drill sizes.

No. and Threads per Inch of Screw or Tap.	Maximum Screw Diameter.	Maximum Tap Diameter.	Nearest Diameter in 64ths.	Body Drill.	Tap Drill.
0×80	.060	.0632	1/16	52	56
1×72	.073	.0765	5/64	5/64	53
2×64	.086	.0898	3/32	42	50
3×56	.099	.1033	7/64	37	47
4×48	.112	.1168	1/8	31	43
5×44	.125	.1301	1/8	29	39
6×40	.138	.1435	9/64	27	35
7×36	.151	.1569	5/32	21	31
8×36	.164	.1699	11/64	11/64	29
9×32	.177	.1835	3/16	13	28
10×30	.190	.1968	13/64	8	24
12×28	.216	.2232	7/32	1	17
14×24	.242	.2500	1/4	1/4	10
16×22	.268	.2765	9/32	J	3
18×20	.294	.3031	5/16	5/16	A
20×20	.320	.3291	21/64	Q	G
22×18	.346	.3559	23/64	U	K
24×16	.372	.3828	3/8	W	19/64
26×16	.398	.4088	13/32	Z	P
28×14	.424	.4359	7/16	7/16	R
30×14	.450	.4619	29/64	15/32	U

Attention. — The body drills given in tables are the nearest commercial drills larger than maximum tap sizes to provide an easy fit. The sizes of the tap drills given are larger than the root diameters of threads for clearance.

Drills designated by parts of an inch are according to Table given in § 450 and Fig. 294. Drills designated by letters are according to the Table of Letter Size Drills, see § 451, and those designated by numbers are according to the Twist Drill and Steel Wire Gage and

Digitized by Google

Table, given in § 452 and Fig. 295. In the absence of these tables the sizes of the body drills required may be obtained by measuring the outside diameter of the tap, and the sizes of tap drills required may be obtained by measuring the root diameter of the tap and allowing a little for clearance.

570. Table of American Society of Mechanical Engineers' special machine screws, taps, and tap drill sizes.

No. and Threads per Inch of Screw or Tap.	Maximum Screw Diameter.	Maximum Tap Diameter.	Nearest Diameter in 64ths.	Body Drill.	Tap Drill.
1×64	.073	.0768	5/64	5/64	54
2×56	.086	.0903	3/32	42	51
3×48	.099	.1038	7/64	37	48
4×40	.112	.1175	1/8	31	45
4×36	.112	.1179	1/8	31	46
5×40	.125	.1305	1/8	29	40
5×36	.125	.1309	1/8	29	42
6×36	.138	.1439	9/64	27	36
6×32	.138	.1445	9/64	27	38
7×32	.151	.1575	5/32	21	32
7×30	.151	.1578	5/32	21	33
8×32	.164	.1705	11/64	11/64	30
8×30	.164	.1708	11/64	11/64	30
9×30	.177	.1838	3/16	13	28
9×24	.177	.1850	3/16	13	30
10×32	.190	.1965	13/64	8	23
10×24	.190	.1980	13/64	8	28
12×24	.216	.2240	7/32	1	19
14×20	.242	.2511	1/4	1/4	14
16×20	.268	.2771	9/32	J	4
18×18	.294	.3039	5/16	5/16	1
20×18	.320	.3299	21/64	Q	F
22×16	.346	.3568	23/64	U	I
24×18	.372	.3819	3/8	W	5/16
26×14	.398	.4099	13/32	Z	O
28×16	.424	.4348	7/16	7/16	S
30×16	.450	.4608	29/64	15/32	V

571. American Screw Company's machine screws. — These screws and taps to correspond are numbered from 0 to 30 and have the Sharp V form of thread. The difference between consecutive sizes is .01316″.

Digitized by Google

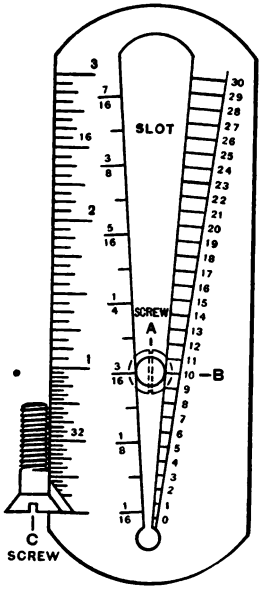


FIG. 372.—AMERICAN SCREW COMPANY'S MACHINE SCREW AND WIRE GAGE.

572. American Screw Company's machine screw and wire gage, Fig. 372, is graduated on one side as shown, on the other as the English wire gage, from 17 to 0000 on the right; and as the American wire gage, 15 to 0000 on the left of the slot. The diameter of screw *A* is 10, shown at *B*. For diameter in inches, see Tables §§ 573, 656. The notch in side of gage facilitates measuring length of screws, as *C*.

Attention. — For practical uses this gage may be utilized also to measure machine screws of the A. S. M. E. Standard, see Tables §§ 569, 570.

573. American Screw Company's machine screws and taps, also body and tap drill sizes.

No. and Threads per Inch of Screw or Tap.	Maximum Screw and Tap Diameter.	Nearest Diameter in 64ths.	Body Drill.	Tap Drill.
2×56	.08416	$\frac{5}{64}$″	42	48
4×36	.11048	$\frac{7}{64}$″	31	41
6×32	.13680	$\frac{9}{64}$″	27	33
8×32	.16312	$\frac{5}{32}$″	18	28
10×24	.18944	$\frac{3}{16}$″	9	23
12×24	.21576	$\frac{7}{32}$″	1	15
14×20	.24208	$\frac{15}{64}$″	$\frac{1}{4}$	10
16×18	.26840	$\frac{17}{64}$″	J	5
18×18	.29472	$\frac{19}{64}$″	N	1
20×16	.32104	$\frac{21}{64}$″	Q	E
22×16	.34736	$\frac{11}{32}$″	T	K
24×16	.37368	$\frac{3}{8}$″	$\frac{3}{8}$	N
26×16	.40000	$\frac{13}{32}$″	$\frac{13}{32}$	P
28×14	.42632	$\frac{27}{64}$″	$\frac{7}{16}$	R
30×14	.45264	$\frac{29}{64}$″	$\frac{29}{64}$	U

Attention. — This table gives the most used sizes of screws and taps from 0 to 30.

Digitized by Google

574. Table of Sharp V-thread taps, $\frac{9}{32}$″ and smaller, in binary fractions with tap drill in numbered sizes (twist drill and wire gage).

Sizes and Threads per Inch of Taps.	Tap Drill.	Sizes and Threads per Inch of Taps.	Tap Drill.
$\frac{1}{16}\times 60$	No. 55	$\frac{3}{16}\times 28$	No. 26
$\frac{5}{64}\times 60$	" 52	$\frac{3}{16}\times 30$	" 23
$\frac{3}{32}\times 48$	" 47	$\frac{3}{16}\times 32$	" 23
$\frac{3}{32}\times 56$	" 46	$\frac{13}{64}\times 24$	" 21
$\frac{3}{32}\times 60$	" 46	$\frac{13}{64}\times 28$	" 20
$\frac{7}{64}\times 32$	" 45	$\frac{13}{64}\times 32$	" 20
$\frac{7}{64}\times 36$	" 44	$\frac{7}{32}\times 22$	" 19
$\frac{7}{64}\times 40$	" 43	$\frac{7}{32}\times 24$	" 18
$\frac{7}{64}\times 44$	" 43	$\frac{7}{32}\times 28$	" 17
$\frac{7}{64}\times 48$	" 42	$\frac{7}{32}\times 30$	" 15
$\frac{1}{8}\times 32$	" 40	$\frac{7}{32}\times 32$	" 13
$\frac{1}{8}\times 36$	" 38	$\frac{15}{64}\times 22$	" 10
$\frac{1}{8}\times 40$	" 37	$\frac{15}{64}\times 24$	" 10
$\frac{1}{8}\times 44$	" 36	$\frac{15}{64}\times 28$	" 9
$\frac{9}{64}\times 30$	" 35	$\frac{15}{64}\times 32$	" 9
$\frac{9}{64}\times 32$	" 32	$\frac{1}{4}\times 20$	" 7
$\frac{9}{64}\times 36$	" 35	$\frac{1}{4}\times 22$	" 5
$\frac{9}{64}\times 40$	" 33	$\frac{1}{4}\times 24$	" 2
$\frac{5}{32}\times 30$	" 31	$\frac{1}{4}\times 32$	" 2
$\frac{5}{32}\times 32$	" 30	$\frac{17}{64}\times 18$	" 4
$\frac{5}{32}\times 36$	" 29	$\frac{17}{64}\times 20$	" 2
$\frac{5}{32}\times 40$	" 29	$\frac{17}{64}\times 24$	" 3
$\frac{11}{64}\times 32$	" 30	$\frac{17}{64}\times 32$	" 1
$\frac{11}{64}\times 36$	" 29	$\frac{9}{32}\times 18$	" 2
$\frac{11}{64}\times 40$	" 28	$\frac{9}{32}\times 20$	" 1
$\frac{3}{16}\times 24$	" 27	$\frac{9}{32}\times 24$	" 1

Attention. — These taps are obtainable commercially, but the screw must be "home made" or obtained by special order.

COUNTERBORING AND COUNTERSINKING.

575. A counterbore, Fig. 373, is a tool used to make a concentric cylindrical enlargement at the end of a hole to

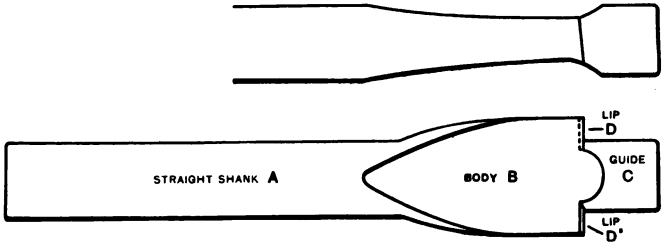


Fig. 373. — Counterbore for Facing Seat for Bolt Head or Screw.

Digitized by Google

receive a bolt head or screw head as in Fig. 374, and to provide an accurate seat, and the process is called *counterboring*. In Fig. 373 a home-made counterbore is shown. Straight shank *A* may be held in a chuck; head or body *B* is slightly larger than head of bolt or screw; guide *C* fits body-drill hole; and *D* and *D'* are the cutting lips which show clearance.

Counterbores with straight or taper shanks are obtainable in sets of three: one with head the size of screw head and with guide the size of body drill; one with head the size of

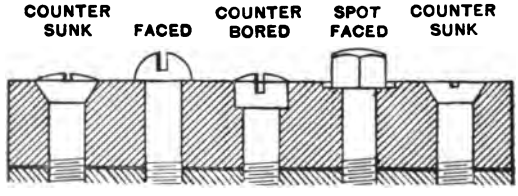


FIG. 374. — COUNTERBORED AND COUNTERSUNK HOLES FOR BOLTS AND SCREW HEADS.

screw head, and guide the size of tap drill; and one with head of body size and guide the size of tap drill to enlarge a tap-drill hole in a cap to body size. They are used similarly to a twist drill and with a speed of from one-half to two-thirds that of twist drills of same diameter of counterbore head. A fine hand feed is generally used for spot facing and deep counterboring.

For rough work a fast speed and a coarse feed may be used, but for fine work use a medium speed and a fine feed.

For deep counterboring power feed may be used of about one-half to two-thirds that used for twist drills, and the counterbore withdrawn occasionally to remove the chips or it will clog and break.

Attention. — A counterbore often glazes the surface of cast iron or hard steel and will not cut unless it is roughed up by indentations with a chisel.

Note. — Cast iron is counterbored dry, but if the guide is a close fit it should be oiled.

Digitized by Google

576. Counterboring a cap for fillister head screw, Fig. 375. Cap *E* is clamped to table *F* by strap *G*, bolt *H*, and block *K*. Commercial counterbore *L* has four cutting edges with clearance on the ends. After the holes are drilled each

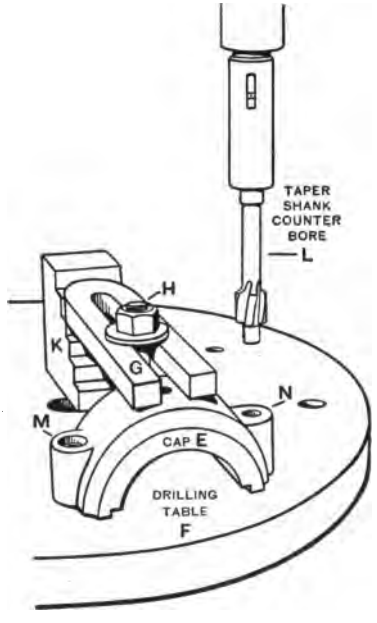


FIG. 375. — COUNTERBORING FOR SCREW HEADS.

hole may be alined with the counterbore and counterbored, or each hole may be counterbored after drilling and before the setting is disturbed. Test depth of counterboring with the screw head. See Fig. 374.

577. Counterbores with separate bodies and guide bushings are obtainable so that various diameters of bodies and guide bushings may be used on the same shank, or stem and guide. They are obtainable in sets for the different sizes of screw heads.

Digitized by Google

578. Counterbore or facing bar with inserted cutter, Fig. 376. — Interchangeable cutters and guides may be used for counterboring or spot facing bosses in the regular way or on the lower side by feeding spindle upward. It consists of

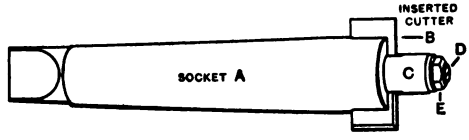


FIG. 376. — COUNTERBORE OR FACING BAR.

socket *A* slotted to receive carbon or high-speed steel cutter *B*. The cutter is centered, and guide bushing *C* is held in place by conical pointed set screw *D*, and lock nut *E*. This tool may be fitted also to the footstock spindle of an engine lathe and used to bore cored work.

A quick method of making a counterbore or facing bar is to drill a hole through a bar, insert a piece of tool steel for cutter, fasten with a set screw, turn and file cutting edges, then harden and temper.

579. A countersink, Fig. 377, is a tool used to make a concentric conical enlargement at the end of a hole to provide

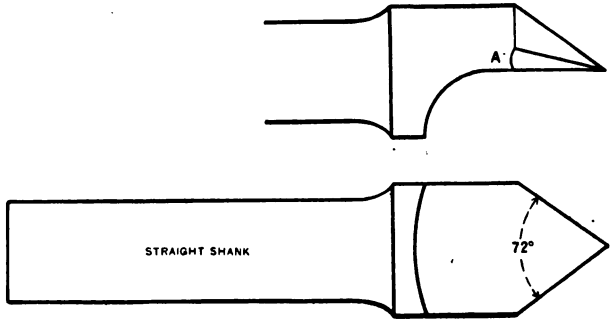


FIG. 377. — COUNTERSINK TO PROVIDE SEAT FOR HEAD OF FLAT-HEAD SCREWS.

a seat for a conical-shaped screw head as in Fig. 374, and the process is called *countersinking*. Countersinks are obtainable

Digitized by Google

with straight or taper shanks, with or without a guide. The head is large enough for several diameters of screw heads. The cutting edge is backed off for clearance at *A*.

The included angle for screw heads, both metal and wood, ranges from 72° to 82°, and this countersink of 72° will answer for this range of angles.

CALCULATING DIAMETER OF BLANK TO MILL SQUARE OR HEXAGONAL.

580. Diameter to turn work to mill or file square is the product of diameter across the flats multiplied by 1.414.

Example. — Fig. 378. What diameter must a piece be to mill square 1¼″ across the flats?

Solution. — 1.250″ × 1.414 = 1.767″ or 1$\frac{25}{32}$″ diameter of blank.

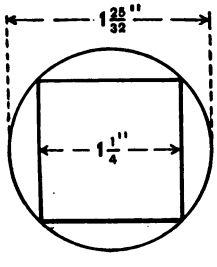


FIG. 378. — DIAGRAM TO TURN CYLINDER TO MILL SQUARE.

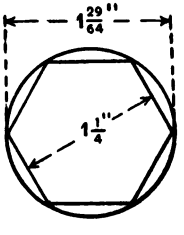


FIG. 379. — DIAGRAM TO TURN CYLINDER TO MILL HEXAGONAL.

581. Diameter to turn work to mill or file hexagonal is the product of the diameter across the flats multiplied by 1.155.

Example. — Fig. 379. What diameter must a piece be to mill hexagonal 1¼″ across the flats?

Solution. — 1.250″ × 1.155 = 1.444″ or 1$\frac{29}{64}$″ diameter of blank.

Digitized by Google

INDEXING IN ENGINE LATHE.

582. To index in engine lathe, Fig. 380. — To file round work square or hexagonal, or to drill diametrically through a shaft, equidistant lines may be drawn on the work to facilitate the operations.

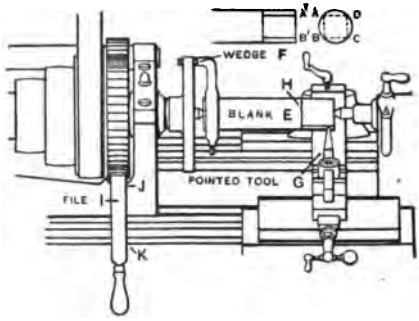


FIG 380. — INDEXING IN THE LATHE.

SCHEDULE OF OPERATIONS.

To divide tap shank circumference into four equal parts, *A*, *B*, *C*, *D*, Fig. 380. Select engine lathe with headstock gear divisible by 4, as $72 \div 4 = 18$. Count headstock gear and mark divisions with chalk. Mount blank *E* on centers with wedge *F* between dog and face plate to prevent back-lash. Use pointed tool *G* to mark line *H* required distance from end.

Place file *I* against under side of chalked tooth; rotate lathe until file touches headstock at *J*.

With the left hand press the handle downward until file touches the bed at *K* and hold it in this position.

With the right hand operating cross feed, move tool to lightly touch work, then change the right hand to long. feed handle and move the carriage to make a line with the tool. For the other lines repeat at the other chalked teeth on gear. Two lines are also shown at *A'*, *B'*.

Digitized by Google

CHAPTER XVIII.

INSIDE CALIPERS AND INSIDE MICROMETERS. BORING AND INSIDE THREADING. SQUARE THREADS. ACME STANDARD OR 29° THREADS. MULTIPLE THREADS. ALINEMENT DRILLING AND TAPPING. ECCENTRIC TURNING.

INSIDE CALIPERS AND INSIDE MICROMETERS.

583. To set inside calipers. Fig. 381. — Hold rule *A* perpendicularly against carriage *B*. Place calipers *C* with point at *D*,

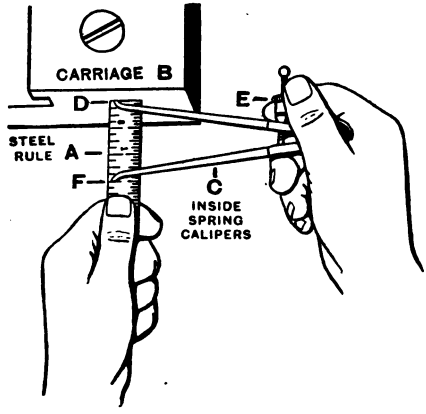


FIG. 381. — SETTING INSIDE CALIPERS.

and adjust nut *E* until other point coincides with middle of line *F*.

Another method is to set calipers to a standard ring gage, or to a hole of the desired diameter in any piece of work.

584. To measure diameters of holes with inside calipers. — Work to be measured may be held in any position on the bench,

Digitized by Google

in vise or chuck. To measure work *G*, Fig. 382, in chuck *H*, set calipers *K* to size and insert point *M* in the lower side of hole, and steady with finger while a gentle effort is made to

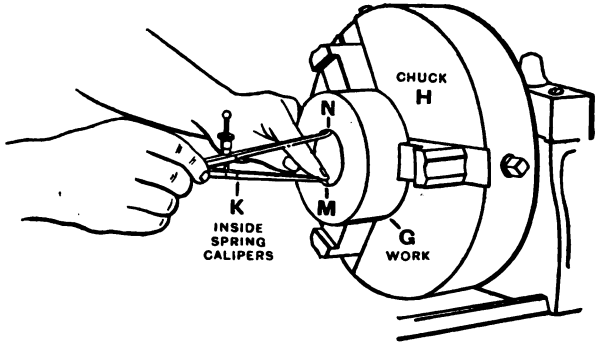


FIG. 382. — MEASURING WITH INSIDE CALIPERS.

nsert point *N*, pivoting calipers on point *M* by raising and lowering the outer end; also move point *N* to right and left to locate maximum diameter.

585. To adjust the tool to bore a hole to diameter to which inside calipers are set. — Take trial cuts and test frequently with calipers. See §§ 588, 589.

586. Small inside micrometer calipers, Figs. 383, 384 and 385, are obtainable in two sizes, one measuring from two-

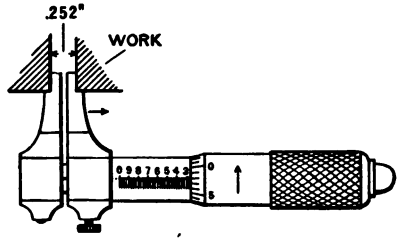


FIG. 383. — MEASURING WITH 1" INSIDE MICROMETER.

tenths of an inch to one inch, and the other from one inch to two inches.

Digitized by Google

Except that the barrels are figured from right to left, they are similar to outside micrometers having 40 threads to the inch.

The reading of the one-inch micrometer in Fig. 383 equals $10 \times .025 = .250'' + .002 = .252''$ or $\frac{1}{4}'' + .002''$.

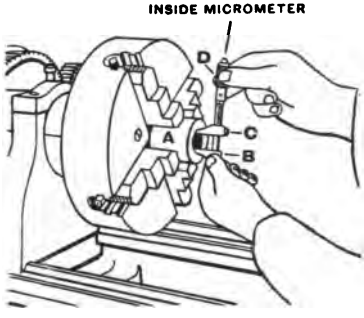


FIG. 384. — MEASURING BORE OF BUSHING.

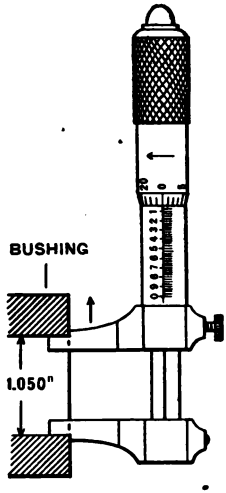


FIG. 385.— MEASURING WITH 2" INSIDE MICROMETER.

In Fig. 385 a two-inch micrometer is shown as used to measure the hole in bushing *A*.

Solid jaw *B* is placed against the lower wall of hole.

Sliding jaw *C* is moved against the upper wall of hole by turning thimble *D* to right. The hole is bored out with a boring tool (see §§ 588, 589) until the two-inch micrometer reads $2 \times .025'' = .050''$.

$1'' + .050'' = 1.050''$ diameter of bore of bushing, as in Fig. 385.

587. Large inside micrometer caliper. — Fig. 386 consists of barrel *A* graduated into 40 divisions to the inch, thimble *B* graduated into 25 divisions, attached to a screw having 40

Digitized by Google

threads to the inch, passing through a nut in end of barrel A. The screw has a movement of half an inch. Measuring point C is fixed to the thimble, but measuring point or rod D is held in chuck E and clamped by nut F and is removable in

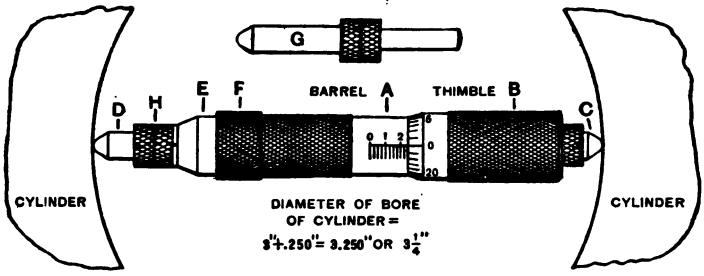


FIG. 386. — MEASURING BORE OF CYLINDER WITH LARGE INSIDE MICROMETER.

order to insert extension rods, a number being supplied varying in length by half an inch ($\frac{1}{2}''$), one being shown at G. Adjustment for wear on rods is provided by adjusting nuts H. Fig. 386 shows how the micrometer is used to measure the bore of a cylinder. The net length of micrometer is 3″ and the reading is $3.200'' + 2 \times .025 = 3.250''$.

BORING AND INSIDE THREADING.

588. Boring tools. — Fig. 387 shows a forged boring or inside turning tool. Cutting edges A and B must be shaped with accuracy. The point is rounded slightly in order to make

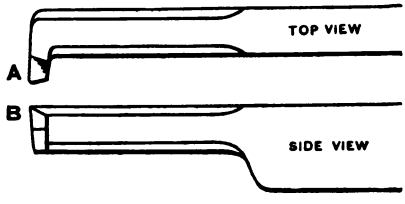


FIG. 387. — FORGED BORING TOOL.

it cut smoothly, and, also, not dull quickly; if rounded too much, the tool will spring away from the cut, or chatter.

Digitized by Google

589. To set and use boring tool in lathe. — Fig. 388.

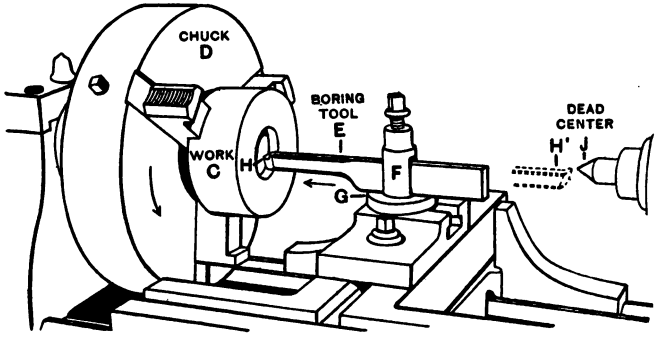


FIG. 388. — BORING IN ENGINE LATHE.

SCHEDULE OF OPERATIONS.

1. True up cored work *C* in chuck *D* and face front end.
2. Set tool *E* height of dead center *J* as at *H'*, reverse tool and post *F* into position parallel to ways of lathe and clamp tightly on *G*.
3. Run tool in length of hole to see that shank clears walls, also chalk top of tool to show length of hole.
4. Rough bore hole and caliper frequently.
5. Take two or three light finishing cuts to leave hole smooth and true. See §§ 583–586.

Attention. — A method sometimes used to bore a straight and smooth hole is to take a light finishing cut inward, then reverse the feed and let the tool cut outward.

Note. — While a boring tool will cut satisfactorily if set at *height* of centers, as *H'*, Fig. 338, still the tool will cut better if set *below* the centers, the amount increasing with the diameter of the hole, as is inversely true with outside turning tools. See §§ 72—74.

This, however, is not always possible, especially in small holes as the size of tool will not allow sufficient clearance and will cause it to ride on the wall of the hole, which must be avoided.

590. Squaring of an inside shoulder with the tool *K*, Fig. 389. *L* is a section taken at *MN*. The rounded point of a boring tool leaves a fillet at the termination of the cut and if

Digitized by Google

a square shoulder is desired an inside squaring tool is used to remove the fillet and square the shoulder by cutting to or from the center.

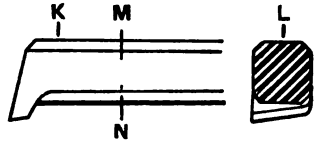


FIG. 389. — INSIDE SQUARING TOOL.

591. Boring holders and cutters are used in the same manner as a forged tool. *A*, Fig. 390, shows holder and double-end cutter rough boring a cored hole, *B*, in work *C*. For inside squaring a special cap is supplied which holds the cutter at an angle of 45° as at *D*. See No. 20, Chart, Fig. 131.

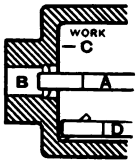


FIG. 390. — BORING AND INSIDE SQUARING WITH HOLDERS AND CUTTERS.

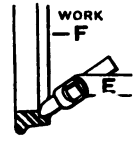


FIG. 391. — BORING AND FACING WITH BENT HOLDER AND CUTTER.

At *E*, Fig. 391, a right-bent outside holder and cutter is shown used as a boring tool and for squaring and facing as at *F*. For boring long holes it is more practical to use drills, boring bars, boring heads, etc.

592. Inside threading tools, United States Standard or Sharp V threads. — Fig. 392 shows point of inside threading tool. It

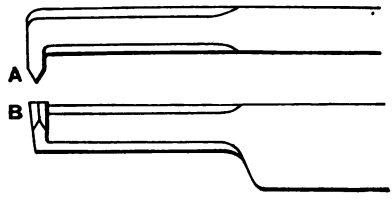


FIG. 392. — INSIDE V-THREADING TOOL.

may be ground as at *A* and *B* for United States Standard or Sharp V threads. It is similar to a boring tool. See No. 34, Chart, Fig. 33.

Digitized by Google

The method of setting an inside United States Standard or Sharp V-threading tool at right angles to the work is shown

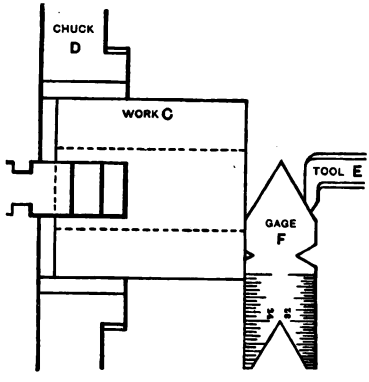


FIG. 393. — SETTING UNITED STATES STANDARD OR SHARP V-THREADING TOOL WITH CENTER GAGE.

in Fig. 393. Work *C* is held in chuck *D* and is bored to size and end rough and finish squared. Tool *E* is then set to gage *F*.

593. To cut an inside thread in lathe. — Fig. 394.

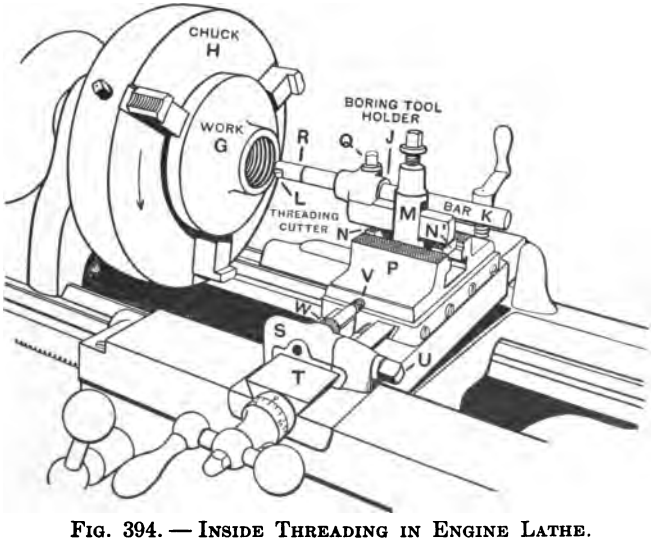


FIG. 394. — INSIDE THREADING IN ENGINE LATHE.

Digitized by Google

SCHEDULE OF OPERATIONS.

1. True up work *G* in chuck *H*, and bore hole about $\frac{1}{64}$″ larger than root diameter of screw that it is to fit, and square end. See Table of Tap Drills, § 538. See Screw Cutting, § 226.

2. Assemble holder *J*, cutter bar *K*, threading tool *L*, which should be ground to fit thread gage, and clamp in tool post *M* supported by blocks *N* and *N′* in tool block *P*.

Adjust tool to height of center, set by gage *F*, Fig. 393, and clamps bar and cutter firmly by screw *Q* and cap *R*.

3. The cutter may be removed, reground and reset to resume its cut by means of cap *R*.

4. Clamp thread stop *S* to slide *T* by screw *U* and adjust feed of tool by rotating nurled head *W* which is fast on screw *V*.

5. Rough thread with cuts from .003″ to .004″ nearly to size. Finish thread with cuts from .001″ to .002″ until thread fits screw, as follows:

Set outside calipers to the outside of the thread, then transfer setting to inside calipers or to a wire filed to fit outside calipers, and pointed at each end more acute than the thread. Cut thread slightly smaller than inside calipers or wire, then test it with the screw.

If work is to fit a lathe spindle or other work that cannot be removed, take chuck and work from lathe, clean, oil, and try on the screw. If it does not fit take another light cut, and so on until desired fit is obtained.

Attention.—If work is cast iron, thread dry; if steel or wrought iron, use lard oil.

Use oil for all materials when fitting to screw or work may seize screw and have to be split off, thereby destroying work and possibly the screw.

594. To finish tap the backing plate of a chuck, Fig. 395. — When cutting an accurate thread such as that in a backing plate, *A*, of a chuck or face plate of a lathe, it is best, if a suitable tap is available, to cut about three-quarters of a full thread with a threading tool, then finish tap as in Fig. 395. Clean and oil thread and tap and place tap wrench *B* on tap *C*, and mount on dead center with tap in thread. Start tap carefully so that it will follow the thread already cut and not split and destroy it. Pull belt *D* by hand and follow tap with dead center with handle *E*. To back out tap, unclamp footstock and run lathe backward by hand or by power. Then square up end *F* and bore out about two threads, as at *G*, to

Digitized by Google

permit screwing plate to shoulder on nose of spindle or on a mandrel to be machined.

595. Interrupted thread tap, Fig. 395. — Instead of using adjustable tap *C*, with regular thread, preferably, use an

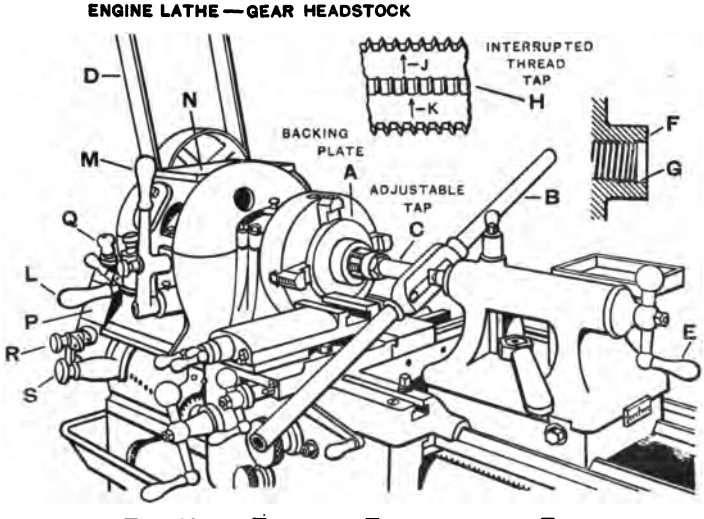


FIG. 395. — FINISHING THREAD WITH A TAP.

interrupted thread tap, as at *H*, which is obtainable. This tap requires less power to drive it. Every other tooth is cut away. The teeth of each land follow in the spaces of the land preceding as shown by arrows *J* and *K*.

596. Engine lathe, gear headstock, Fig. 395, shows an engine lathe equipped with a gear speed change located in the head in place of the cone pulley. The different speeds and positions of the levers *L* and *M* to obtain them are given in a table at *N*.

Attention. — Some all-gear headstock engine lathes have a variable speed countershaft to give a still greater variety of speeds.

This lathe is also equipped with a rapid change-gear mechanism for feeds and threads. In a table at *P* are given the different threads, and positions of the levers *Q*, *R*, and

Digitized by Google

S, to obtain them. The feed is usually seven times threads per inch.

597. To cut right inside thread to shoulder. — Fig. 396.

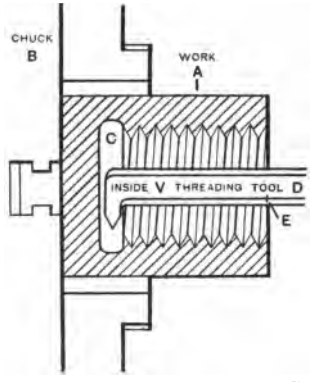


FIG. 396. — CUTTING INSIDE UNITED STATES STANDARD OR SHARP V THREAD TO A SHOULDER.

SCHEDULE OF OPERATIONS.

1. Work *A*, held in chuck *B*, is bored to correct diameter and depth.	3. Mark line *E* with chalk to indicate when tool *D* reaches groove.
2. Cut groove *C* to full diameter of thread with inside forming tool.	4. Cut thread to desired depth.

598. To cut a left inside thread to a shoulder, or any portion of the hole less than its entire length, cut groove as at *C*, Fig. 396, from which start tool outward, and make a mark at *E* to know when to adjust the tool forward into groove preparatory to starting to cut outward.

SQUARE THREADS.

599. Square threads, Fig. 397, right or left, are used for screws to transmit motion, as the cross feed and lead screws of an engine lathe, valve stems, presses, rock drill feed screws, etc. It cannot be cut successfully with dies, or milled with a thread milling machine. See Acme Standard or 29° Threads, § 612.

Digitized by Google

The thickness of thread and width of space are each nominally one-half the pitch. The depth is one-half pitch plus the clearance. The fit is on the sides of the thread with clearance top and bottom. A larger clearance is advisable for large diameters and coarse pitchers.

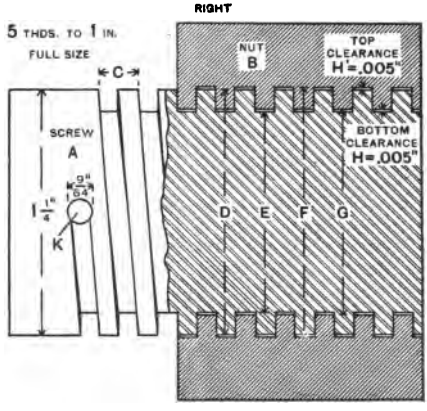


FIG. 397. — SECTIONAL VIEW OF SQUARE THREAD SCREW AND NUT.

NAMES OF PRINCIPAL PARTS OF SQUARE THREAD.

C. — Pitch.	*G.* — Bore of nut.
D. — Diameter standard.	*H* and *H'*, clearances; same for all pitches. *H* made by cutting the thread .005″ deeper than $\frac{1}{2}$ *P*.
E. — Root diameter (which is also root diameter of tap).	*H'* made by making tap. 01″ larger in diameter than *D*.
F. — Diameter of bottom of nut (which is also diameter of tap).	

600. To obtain parts of Square thread. —

Width of tool for screw thread =

$$\frac{1}{\text{No. of threads per inch}} \div 2.$$

Width of tool for tap thread =

$$\frac{1}{\text{No. of threads per inch}} \div 2 -$$

Digitized by Google

.0005″ for each linear inch of nut for shrinkage of tap.

Diameter of screw = any

Diameter of tap = diameter of screw + .010″

Diameter of screw or tap at root of thread = diameter of screw —

$$\frac{1}{\text{No. threads per inch}} + .010''.$$

Diameter to bore nut = diameter of screw —

$$\frac{1}{\text{No. of threads per inch}}.$$

Clearance = .005″, top and bottom of thread.

601. Table of Square threads. — While any pitch may be assumed, it is best, when it will answer the purpose, to use whole numbers of threads per inch as near as possible to three-quarters of the United States Standard thread. See §221.

Diameter.	Threads Per Inch.	Diameter.	Threads Per Inch.
½″	10	1″	6
⅝″	9	1¼″	5
¾″	8	1½″	4
⅞″	7	2″	3

602. Square threading Tool, Fig. 398.

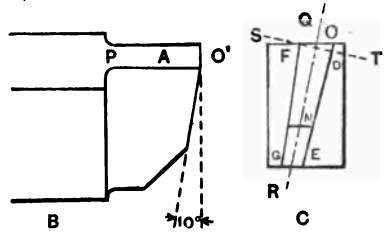


Fig. 398. — Outside Square Threading Tool.

A, top face, *B*, side view partially rotated, and *C*, end view. *DE* and *FG* inclined to line *QR* to give clearance.

Digitized by Google

TO FIND INCLINATION OF THREAD AND TO FILE TOOL.

SCHEDULE OF OPERATIONS.

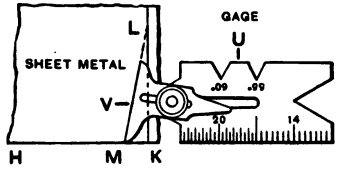


FIG. 399. — MAKING A TEMPLET AND SETTING GAGE TO ANGLE OF INCLINATION FOR SQUARE THREADING TOOL.

Inclination. Fig. 399.

1. File line *HK* true.
2. Draw *KL* at 90° to *HK*.
3. Make *KL* equal to root circumferences of thread.
4. Make *MK* equal to pitch.
5. File to *LM* which gives inclination of thread and use as a templet to test tool or omit filing and set angle gage *U*, Fig. 399, to angle *LM* as at *V*.

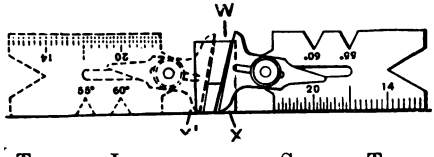


FIG. 400. — TESTING INCLINATION OF SQUARE THREADING TOOL WITH ANGLE GAGE.

To File Tool. Fig. 400.

1. File bottom of tool flat, and end square as in Fig. 398. File sides with an 8″ or 10″ hand smooth file and 8″ dead smooth file to give inclination, clearance, and size.
2. Test tool with gage *U* as at *X* and *X'*, Fig. 400, or with sheet metal templet *V*, as in Fig. 399.
3. Measure with micrometer calipers.
4. Harden and temper to a straw color.
5. Grind on end and a little on top.

Do not grind on sides.

Attention. — The amount of inclination varies with different pitches and diameters, but with a generous side clearance one tool will do for several diameters. The tool is parallel from *O'* to *P*, Fig. 398.

Digitized by Google

For very coarse pitches tool is narrower at P than at O'. For fine pitches top face A is ground horizontal; for coarse threads it is ground on line ST at right angles to line QR.

Note. — The tool may be forged, and ground to the proper angle with a universal tool grinder, see *Elements of Machine Work*, then hardened and tempered, reground and the filing omitted.

603. An outside Square threading holder and cutter is shown at A and B, Fig. 401. Holder C supports cutter D. Clamp E and bolt F fasten cutter to holder and hold different widths of

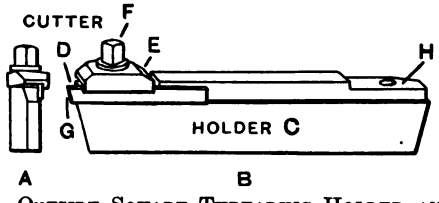


FIG. 401. — OUTSIDE SQUARE THREADING HOLDER AND CUTTER.

cutters. Cutter is ground upon end G only. Roughing and finishing cutters are used for coarse threads. End H is for left threads.

604. Method of setting outside Square threading tool. — Fasten tool A, Fig. 402, in tool-post lightly and adjust to height of dead center. Mount screw blank B on centers.

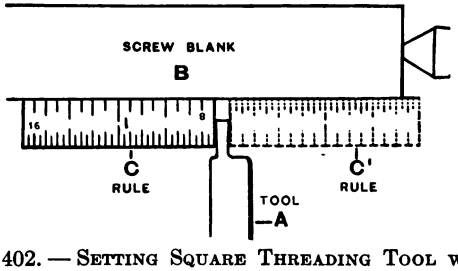


FIG. 402. — SETTING SQUARE THREADING TOOL WITH STEEL RULE.

Place steel rule C against blank parallel to axis and rap tool until parallel to end of rule, testing from both sides as at C and C'. Fasten tool firmly.

Digitized by Google

605. An inside Square threading tool, Fig. 403, is used for cutting inside threads. The blade *AB* is inclined, shaped and sized to suit thread to be cut in the same manner as the outside Square threading tool, Fig. 400, § 602. Inside

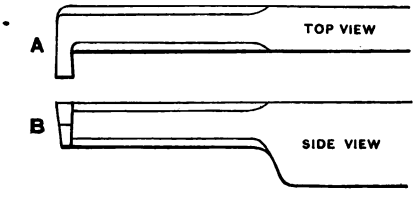


FIG. 403. — INSIDE SQUARE THREADING TOOL.

Square threads are cut also by inserting properly shaped cutters in boring tool holders. See No. 23, Chart, Fig. 131.

606. Method of setting inside Square threading tool. — Nut blank *C*, Fig. 403, is held in chuck *D*. End *E* is faced and hole bored to size. Place tool *G* in tool-post and adjust to height of dead center (see No. 2, §589) and fasten lightly

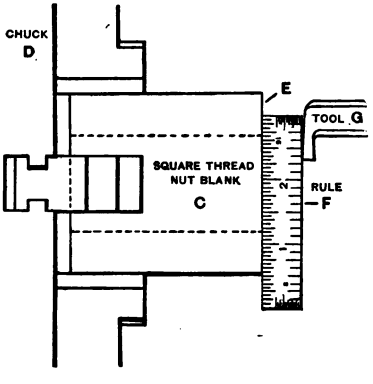


FIG. 404. — SETTING INSIDE SQUARE THREADING TOOL.

in approximate position. Hold rule *F* against end *E* and rap tool until blade of tool is parallel with rule. Then clamp firmly.

Digitized by Google

607. Roughing tool. — For Square threads, five pitch or coarser, use a roughing tool .010″ less in width, *O*, Fig. 398, than finishing tool.

608. Square thread tap $1\frac{1}{4}$″ diameter, 5 threads to 1″, Fig. 405, is used to tap both loose and fixed nuts. Loose nuts are usually rough threaded in the engine lathe with an inside rough Square threading tool, Fig. 403, then finish threaded with one or more taps.

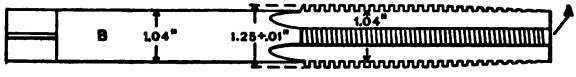


FIG. 405. — SQUARE THREAD TAP, $1\frac{1}{4} \times 5$.

For fixed nuts such as parts of machine frames, one or two roughing taps are used, followed with the finishing tap.

They are made without a leader as at *A*, Fig. 405, or with plain or threaded leaders, *A*, Fig. 413. See Alinement Drilling and Tapping, § 626.

The diameter is made one-hundredth of an inch larger than the screw for clearance. The root diameter is the same as the screw. See § 600.

Shank *B* is made about one-hundredth of an inch smaller than the bore of the nut.

These taps cannot be obtained commercially, but must be specially made.

609. To cut a Square thread screw, Fig. 406.

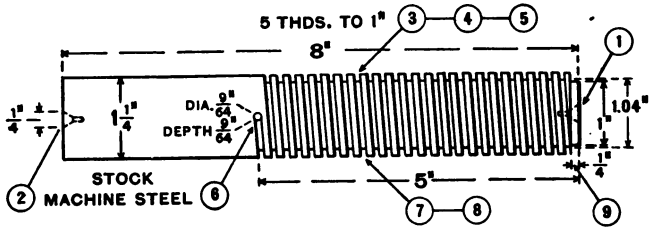


FIG. 406. — SCHEDULE DRAWING OF SQUARE THREAD SCREW.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.

Preparing screw blank. Threading.
Stock, machine steel $\frac{1}{16}''$ large.
True live center.
Set dead center in accurate alinement.
Use high-speed steel cutting tools. See *Exception* p. 59.

Operations.	Machines, Speeds, Feeds.	Tools.
Center.	Centering machine.	$\frac{3}{32}''$ drill, 60° countersink, lard oil.
Rough square, (1), (2).	Engine lathe 12″ to 16″. 2d or 3d speed, or 50 F.P.M. Hand feed.	Dog, rule, calipers, side tool, 35° rake.
Recenter.	Speed lathe, drill 4th speed, countersink 3d speed.	
Finish square, (1), (2).	Engine lathe, 3d or 4th speed or 80 F.P.M. Hand feed.	
Rough turn $\frac{1}{64}''$ large, (3), one cut.	1st speed, or 40 F.P.M. Medium power feed — 80 to 1″.	Diamond-point tool, 35° rake or holder and cutter, calipers, rule.
Finish turn 1.25″ + .004″, (4).	3d speed, or 60 F.P.M. Fine power feed — 140 to 1″.	Copper under set screw of dog, diamond-point tool or holder and cutter, 35° rake, micrometer.
File, (5).	4th or 5th speed, or 175 F.P.M.	8″ or 10″ mill bastard file.
Polish.	Speed lathe, highest speed.	60 and 90 emery cloth, polishing clamps.
Or rough turn .01″ large, and grind, after threading.	See Cylindrical Grinding Machine, § 379.	
Drill hole, (6).	Speed lathe, drill chuck, 3d or 4th speed, or 1000 R.P.M.	$\frac{9}{64}''$ straight shank twist drill V center, depth gage, lard oil.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.
Concluded.

Operations.	Machines, Speeds, Feeds.	Tools.
Set Square thread tool (see § 60.) and thread stop, arrange lathe for 5 threads. Pull belt downward to take up back lash, loosen set screw of dog and adjust shaft until tool terminates in hole, (6). Tighten set screw, rough thread to 1.04″ + .01″, (7), twenty cuts .005″ each. Depth of thread .100″.	Engine lathe. 1st speed, or 25 F.P.M.	Grind forged Square, thread roughing tool, width .090″. See Fig. 398, or use holder and cutter, see Fig. 401, calipers, rule. Lubricate freely with lard oil.
Set finishing tool to cut on both sides of groove by loosening dog, adjusting shaft and testing cut at end thread. Tighten set screw. Finish thread to 1.04″, (8), twenty cuts .005″, two cuts, .002″, one cut .001″. Depth of thread .105″.	1st speed, or 25 F.P.M.	Forged finishing tool width .100″ + .002″ for fit, calipers file, harden and temper and grind or use holder and cutter, calipers, rule, lard oil.
Turn off thick end thread, (9), and smooth thread with file.	2d or 3d speed, or 50 F.P.M. Hand feed.	Diamond-point and side tools, or holder and cutter, 8″ or 10″ mill bastard file.
File top of threads to remove burr.	4th or 5th speed, or 175 F.P.M.	8″ or 10″ mill bastard file.
File sides slightly.	Slow speed for filing sides of thread.	Warding file.

Attention. — Terminate each cut as follows: stop lathe when tool is $\frac{1}{2}$ or $\frac{1}{3}$ revolution from hole, then carefully pull belt to continue cut almost to hole and end the cut by moving tail of dog in slot of face plate.

Note. — Roughing tool may be removed, ground, and reset if necessary, but not the finishing tool.

Digitized by Google

610. To make a Square thread nut, Fig. 407.

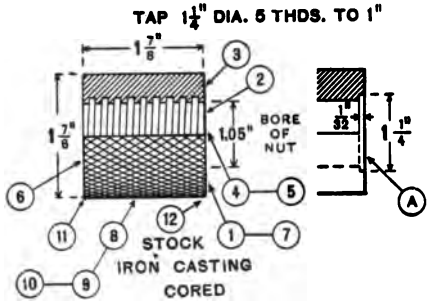


FIG. 407. — SCHEDULE DRAWING OF SQUARE THREAD NUT.

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.

Preparing nut blank. Rough threading. Tapping.
Stock, iron casting, cored.
Use high-speed steel cutting tools. See *Exception*, p. 59.

OPERATIONS.	MACHINES, SPEEDS, FEEDS.	TOOLS.
Mount in chuck, true up and clamp hard in chuck.	Engine lathe 12″ to 16″. 3d speed, or 115 F.P.M.	Independent chuck, chalk.
Rough square, (1), one or two cuts.	2d or 3d speed, or 40 F.P.M. Hand or power feed.	Round-nose tool, or holder and cutter, 15° rake.
Finish square, (1), one or two cuts.	3d or 4th speed, or 60 F.P.M.	Facing tool or holder and cutter, 15° rake.
Rough bore hole to about 1.03″, (2), two or three cuts.	1st or 2d speed, or 40 F.P.M. Medium power feed — 80 to 1″.	Boring tool, see § 588. Inside calipers, rule.
Finish bore hole, (2), two or three cuts.	3d speed, or 60 F.P.M. fine power feed — 140 to 1″.	
Set inside Square thread tool (see Fig. 404), and cut recess for improvised gage 1/32″ × 1¼″, (3). See (A).	2d speed or 50 F.P.M. Hand feed.	Forged Square thread tool, width .090″, or holder and cutter, inside calipers, rule.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.
Continued.

OPERATIONS.	MACHINES, SPEEDS, FEEDS.	TOOLS.
Arrange lathe for 5 threads with thread stop reversed. Rough thread to $1\frac{1}{4}''$ the diameter of gage *A*, (4), fifteen cuts, .006″ each, five cuts, .002″ each. Depth of thread, .100″.	1st speed, or 30 F.P.M.	Without oil.
Start tap in lathe, pull belt downward and follow with dead center. Be sure that tap follows thread or it may ream. Remove nut and tap to reaming stand or vise, and finish tapping, (5).	Reaming stand or vise and grooved wooden jaws.	$1\frac{1}{4}''\times 5$ Square thread, tap and tap wrench, lubricate tap freely with lard oil.
Mount nut on nut mandrel and rough square, (6), one or two cuts.	2d or 3d speed, or 40 F.P.M.	$1\frac{1}{4}''\times 5$ Square thread nut mandrel, dog, round-nose tool, or holder and cutter, calipers, rule.
Finish square, (6), one or two cuts.	3d or 4th speed, or 60 F.P.M.	Facing tool or holder and cutter.
Reverse nut and square to length to remove recess, (7), two or three cuts.	2d or 3d speed or 60 F.P.M.	Round-nose tool or holder and cutter, and facing tools, calipers, rule.
Rough turn, (8), one or two cuts.	1st or 2d speed, or 35 F.P.M. Medium power feed — 80 to 1″.	Diamond-point tool or holder and cutter, 15° rake, calipers, rule.
Finish turn, (8), one cut.	2d or 3d speed, or 50 F.P.M.	Diamond-point or round-nose tool, or holder and cutter, calipers, rule.
File, (9).	4th speed, or 175 F.P.M.	8″ or 10″ mill bastard file.
Nurl, two to four times, (10). See Machine Nurling, § 370.	1st or 2d speed, or 35 F.P.M. Medium power feed — 80 to 1″.	Machine nurling tool, medium pitch, oil.
File corners slightly to remove burr, (11), (12).	4th speed or 175 F.P.M.	8″ or 10″ mill bastard file.

See *Attention* and *Note*, p. 3 5.

Digitized by Google

Attention. — In the absence of a tap an inside tool may be made one-half pitch × .001″ to finish the thread, but the thread will not be as smooth or the fit of screw and nut as good.

Note. — The nut in Fig. 407 is nurled for convenience in handling as a problem. For practical styles of nuts to transmit motion, see § 626.

611. To fit screw to nut. Figs. 406 and 407.

SCHEDULE OF OPERATIONS.

1. Ascertain if thread binds on top or bottom by testing with calipers and comparing with tap, Fig. 405; if so, file top of threads on screw or cut thread deeper.
2. Hold nut in grooved wooden jaws in vise. Oil screw, grasp dog with both hands and force in with hand pressure.
3. If need be, file side of threads with a warding file.
4. After screw is fitted to nut, polish top of thread with 90 emery and clamps.

Attention. — To file sides of a right thread, preferably, run lathe backward at a slow speed, reverse file and file toward footstock.

ACME STANDARD OR 29° THREADS.

612. Acme Standard or 29° threads, Fig. 408, right or left, are used for screws to transmit motion, as on lead screws, feed

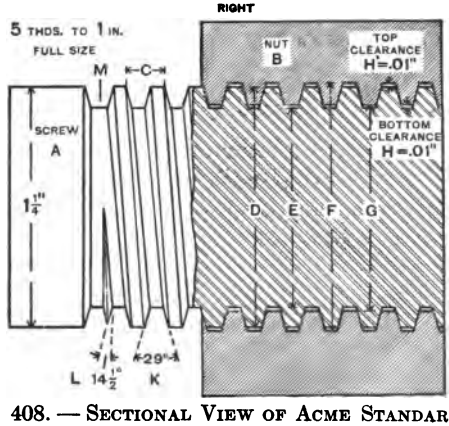


FIG. 408. — SECTIONAL VIEW OF ACME STANDARD OR 29° THREAD SCREW AND NUT.

screws, elevating screws, valve stems, presses, rock drills, etc., and it is displacing the Square thread for other purposes

Digitized by Google

because it can be more readily cut in the lathe, and also successfully and rapidly cut with dies, with bolt cutters and turret lathes, and milled with thread milling machines.

For Thread Milling Machine, see *Advanced Machine Work*. The depth of the thread is equal to one-half the pitch plus .01″ for clearance. The fit is on the sides with clearance on top and bottom.

NAMES OF PRINCIPAL PARTS OF 29° THREAD.

C — Pitch.	H and H' — Clearances, same for all pitches. (H obtained by cutting thread .01″ deeper than $\frac{1}{2}P$; H', by making tap .02″ larger than diameter of screw.)
D — Diameter, standard.	K — Included angle 29°.
E — Root diameter (which is also root diameter of tap).	L — Side angle $14\frac{1}{2}°$.
F — Diameter at bottom of thread on nut (which is also diameter of tap).	
G — Bore of nut.	

613. To obtain parts of thread. —

Width of point of tool for screw or tap thread =

$$\frac{.3707}{\text{No. threads per inch}} - .0052.$$

Width of point of screw or nut thread =

$$\frac{.3707}{\text{No. threads per inch}}.$$

Diameter of tap = diameter of screw + .020.

Diameter of screw = any

Diameter of tap or screw at root of thread = diameter of screw $- \frac{1}{\text{No. threads per inch}} + .020''$.

Depth of thread $= \frac{1}{2 \times \text{No. threads per inch}} + .010.$

Diameter to bore nut = diameter of screw —

$$\frac{1}{\text{No. threads per inch}}.$$

Clearance = .01″ top and bottom of thread.

Digitized by Google

TABLE OF THREAD PARTS FOR ACME STANDARD OR 29° THREAD.

Pitch.	No. of Threads per Inch.	Depth of Thread.	Width at Top of Thread.	Width at Bottom of Thr'd.	Space at Top of Thread.	Thickness at Root of Thread.
2	½	1.010	.7414	.7362	1.2586	1.2637
1⅞	1 5/15	.9475	.6950	.6897	1.1799	1.1850
1¾	4/7	.8850	.6487	.6435	1.1012	1.1064
1⅝	1 3/13	.8225	.6025	.5973	1.0226	1.0277
1½	⅔	.7600	.5560	.5508	.9439	.9491
1 7/16	2 6/23	.7287	.5329	.5277	.9046	.9097
1⅜	1 1/11	.6975	.5097	.5045	.8652	.8704
1 5/16	2 1/21	.6662	.4865	.4813	.8259	.8311
1¼	⅖	.635	.4633	.4581	.7866	.7918
1 3/16	1 6/19	.6037	.4402	.4350	.7472	.7525
1⅛	8/9	.5725	.4170	.4118	.7079	.7131
1 1/16	1 6/17	.5412	.3938	.3886	.6686	.6739
1	1	.510	.3707	.3655	.6293	.6345
15/16	1 1/15	.4787	.3476	.3424	.5898	.5950
⅞	1 1/7	.4475	.3243	.3191	.5506	.5558
13/16	1 3/13	.4162	.3012	.2960	.5112	.5164
¾	1⅓	.385	.2780	.2728	.4720	.4772
11/16	1 5/11	.3537	.2548	.2496	.4327	.4379
⅝	1⅐	.3433	.2471	.2419	.4194	.4246
9/16	1⅖	.3225	.2316	.2264	.3934	.3986
½	1 7/9	.2912	.2085	.2033	.3539	.3591
7/16	2	.260	.1853	.1801	.3147	.3199
⅜	2⅐	.2287	.1622	.1570	.2752	.2804
5/16	2½	.210	.1482	.1430	.2518	.2570
⅛	2⅞	.1975	.1390	.1338	.2359	.2411
⅓	3	.1766	.1235	.1183	.2098	.2150
5/16	3⅕	.1662	.1158	.1106	.1966	.2018
¼	3½	.1528	.1059	.1007	.1797	.1849
¼	4	.1350	.0927	.0875	.1573	.1625
⅛	4½	.1211	.0824	.0772	.1398	.1450
⅕	5	.110	.0741	.0689	.1259	.1311
3/16	5⅓	.1037	.0695	.0643	.1179	.1232
⅛	6	.0933	.0617	.0565	.1049	.1101
⅛	7	.0814	.0530	.0478	.0899	.0951
⅛	8	.0725	.0463	.0411	.0787	.0839
⅛	9	.0655	.0413	.0361	.0699	.0751
1/10	10	.060	.0371	.0319	.0629	.0681
1/16	16	.0412	.0232	.0180	.0392	.0444

Digitized by Google

614. Table of Acme Standard or 29° threads. — While any pitch may be assumed, it is best to use whole numbers of threads per inch as near as possible to three-quarters of the United States Standard. See § 221.

Diameter.	Threads Per Inch.	Diameter.	Threads Per Inch.
½″	10	1″	6
⅝″	9	1¼″	5
¾″	8	1½″	4
⅞″	7	2″	3

615. Acme Standard or 29° threading tool, Fig. 409. *A* is top face, *B* and *C* are side and end views. For method of finding inclination of thread, see Square Threading Tool, § 602.

TO FILE OR GRIND TOOL.

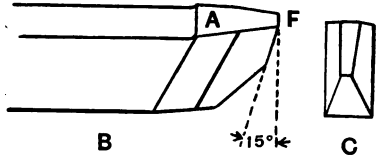


Fig. 409. — Acme Standard or 29° Threading Tool.

SCHEDULE OF OPERATIONS.

1. File bottom and end of tool.
2. File sides to fit gage *D* as at *E*, Fig. 410.
3. Harden and temper to a straw color.
4. Grind end until point *F*, Fig. 409, will fit notch in gage as at *G*, Fig. 410, the desired pitch.
5. Grind top face, *A*, Fig. 409, slightly.

Attention. — For fine pitches, use same tool for roughing and finishing. For 5 pitch or coarser, rough with Square threading tool .01″ narrower than point of 29° tool. In case of very coarse pitches, cut a square groove, then with right and left side tool cut down sides of thread, after which use finishing tool of desired shape. A compound rest is often used for coarse pitches.

Note. — The tool may be forged and ground to the proper angle with a universal tool grinder (see *Elements of Machine Work*), then hardened and tempered, reground, and the filing omitted. The thread is also cut with 29° threading holder and cutter. See No. 16, Chart, Fig. 131.

Digitized by Google

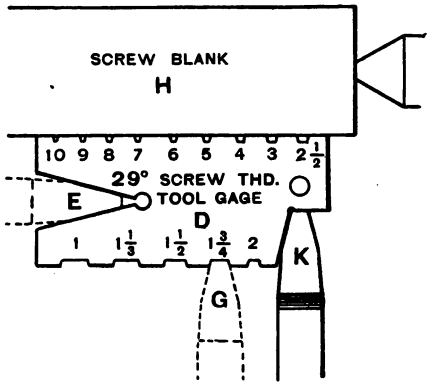


FIG. 410. — SETTING TOOL TO CUT ACME STANDARD OR 29° THREAD.

616. Method of setting outside Acme Standard or 29° threading tool. — Fasten tool *K*, Fig. 410, in tool post lightly and adjust to height of dead center. Mount screw blank *H* on centers. Place gage *D* against screw blank *H* parallel to axis and rap tool until angle of tool fits angle of gage. Then fasten tool firmly.

617. An inside Acme Standard or 29° threading tool. — Fig. 411 is used for cutting inside threads. The blade *AB* is inclined, shaped and ground to suit thread to be cut in the same

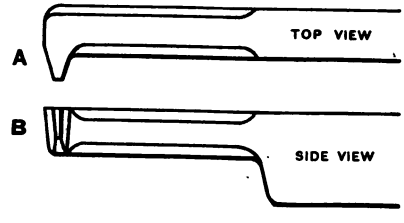


FIG. 411. — INSIDE ACME STANDARD OR 29° THREADING TOOL.

manner as the outside Acme Standard or 29° threading tool, Fig. 409, § 615.

Inside 29° threads are cut also by inserting properly shaped cutters in boring tool holders. See No. 24, Chart, Fig. 131.

Digitized by Google

618. Method of setting inside Acme Standard or 29° threading tool. — Nut blank *C*, Fig. 412, is held in chuck *D*. End *E* is faced and hole bored to size. Place tool *F* in tool post

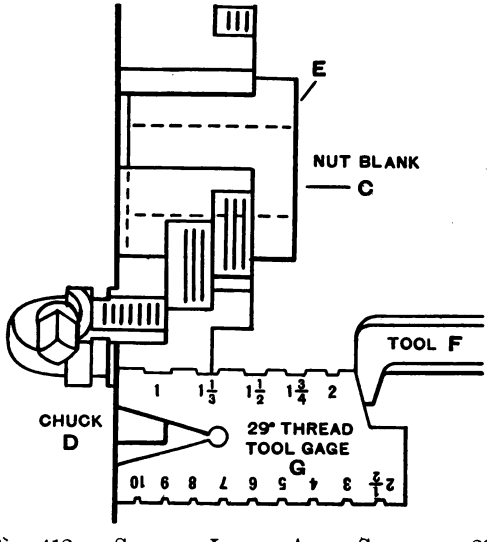


FIG. 412. — SETTING INSIDE ACME STANDARD 29° THREADING TOOL.

and adjust to height of dead center (see No. 2, § 589) and fasten lightly in approximate position. Hold gage *G* against face of chuck *D* and rap tool until angle of tool fits angle of gage. Then fasten tool securely.

619. Acme Standard or 29° thread tap 1¼″ in diameter, 5 threads to 1, Fig. 413, is used to tap both loose and fixed nuts.

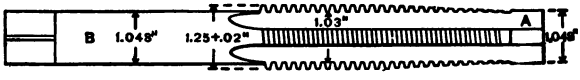


FIG. 413.— ACME STANDARD OR 29° THREAD TAP 1¼ × 5.

Loose nuts are usually rough threaded in the engine lathe with an inside threading tool, Fig. 412, to about ¾ of a full thread, then finish threaded with a tap.

Digitized by Google

For fixed nuts, such as parts of machine frames, one or two roughing taps are used, followed by the finishing tap, preferably of type shown at *H*, Fig. 395. These taps are made with leaders as at *A* or without leaders, *A*, Fig. 405. See Alinement Drilling and Tapping, § 626. The diameter is made two-hundredths of an inch larger than the screw for clearance. See § 613. The root diameter of tap is the same as screw.

Leader *A* and shank *B* are two-thousandths of an inch smaller than bore of nut, and the leader may be used as a gage to test the bore of nut. These taps cannot be obtained commercially but must be specially made.

620. To cut Acme Standard or 29° thread screw, Fig. 414.

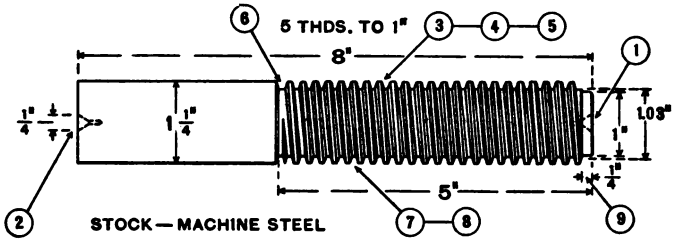


FIG. 414. — SCHEDULE DRAWING OF ACME STANDARD OR 29° THREAD SCREW.

SCHEDULE OF OPERATIONS, MACHINES, AND TOOLS.

Stock, machine steel $\frac{1}{16}'$ large.
True live center.
Set dead center in accurate alinement.

OPERATIONS.	MACHINES, SPEEDS, FEEDS.	TOOLS.
Center.	Centering machine.	$\frac{3}{32}''$ drill, 60° countersink, lard oil.
Rough square, (1), (2).	Engine lathe 12″ to 16″. 2d or 3d speed, or 50 F.P.M. Hand feed.	Dog, rule, calipers, side tool, 35° rake.
Recenter.	Speed lathe, drill 4th speed; countersink, 3d speed.	

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.
Continued.

Operations.	Machines, Speeds, Feeds.	Tools.
Finish square, (1), (2).	Engine lathe, 3d or 4th speed, or 80 F.P.M.	
Rough turn $\frac{1}{64}$″ large, (3), one cut.	1st speed, or 40 F.P.M., medium power feed — 80 to 1″.	Diamond-point tool or holder and cutter, 35° rake, calipers, rule.
Finish turn 1.25″ + .004″, (4).	3d speed, or 60 F.P.M., fine power feed — 140 to 1″.	Copper under set screw of dog, diamond-point tool or holder and cutter, 35° rake, micrometer.
File, (5).	4th or 5th speed, or 175 F.P.M.	8″ or 10″ mill bastard file.
Polish.	Speed lathe, highest speed.	60 and 90 emery cloth, polishing clamps.
Cut groove, (6), to root diameter of thread, 1.03″.	Engine lathe, 1st or 2d speed, or 25 F.P.M. Hand feed.	29° grooving tool, calipers, rule, lard oil.
Set roughing tool and thread stop, arrange lathe for 5 threads. Pull belt downward to take up backlash, loosen set screw of dog and adjust shaft until tool terminates in groove, (6). Tighten set screw, rough thread to 1.03″ + .02″, (7), twenty cuts .005″ each. Depth of thread, .100″.	Engine lathe, 1st speed, or 30 F.P.M.	Grind forged Square thread tool — width .060″, or use holder and cutter, calipers, rule; lubricate freely with lard oil.
Set finishing tool to cut on both sides of groove by taking up backlash, loosening dog, adjusting shaft and testing cut at end thread. Finish thread, (8), twenty cuts of .005″ each, two cuts of .002″ each. Then take 1 cut .001″, clean, oil, and test and repeat until screw fits nut. Depth of thread .110″. See (8).	1st speed, or 25 F.P.M.	Forged 29° finishing tool, ground to fit angle and notch 5 on gage, Fig. 410, or use holder and cutter, calipers, rule, lard oil.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.
Concluded.

Operations.	Machines, Speeds, Feeds.	Tools.
Turn off end of thread, (9), and smooth off thread with file.	2d or 3d speed, or 50 F.P.M. Hand feed.	Diamond-point and side tools, or holder and cutter, 8″ or 10″ mill bastard file.
File top of threads to remove burr.		8″ or 10″ mill bastard file.
File sides of thread slightly if needed to make the fit easier.	Slow speed for filing sides of thread.	Warding file.
Polish tops of threads.	Speed lathe, highest speed.	90 emery, polishing clamps.

Attention. — Terminate each cut as follows: stop lathe when tool is $\frac{1}{2}$ or $\frac{1}{3}$ a revolution from groove, then carefully pull belt to continue cut to groove.

A hole instead of a groove is sometimes used in which to terminate the cut. See K, Fig. 397.

For fine pitches a groove, or hole, is sometimes omitted and a tapering termination used the same as United States Standard and Sharp V threads. See Fig. 157.

Note. — Roughing tool may be removed, ground, and reset, if necessary, but it is best not to remove the finishing tool until thread is completed.

621. To make an Acme Standard 29° thread nut, Fig. 415.

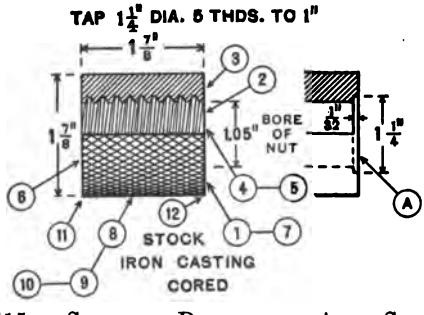


Fig. 415. — Schedule Drawing of Acme Standard or 29° Thread Nut.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.

Preparing nut blank. Rough threading. Tapping.
Stock, iron casting, cored.
Use high-speed steel cutting tools. See *Exception*, p. 59.

Operations.	Machines, Speeds, Feeds.	Tools.
Mount in chuck, true up, and clamp hard in chuck.	Engine lathe 12″ to 16″. 3d speed, or 115 F.P.M.	Independent chuck. Chalk.
Rough square, (1), one or two cuts.	2d or 3d speed, or 40 F.P.M. Hand or power feed.	Round-nose tool or holder and cutter, 15° rake.
Finish square, (1), one or two cuts.	3d or 4th speed, or 60 F.P.M.	Facing tool or holder and cutter, 15° rake.
Rough bore hole to about 1.03″, (2), two or three cuts.	1st or 2d speed, or 40 F.P.M, medium power feed — 80 to 1″.	Boring tool, § 588. Inside calipers, rule.
Finish bore hole, (2), two or three cuts.	3d speed, or 60 F.P.M.	
Set inside 29° thread tool, Fig. 412, cut recess for improvised gage $\frac{1}{32}'' \times 1\frac{1}{4}''$, (3). See (*A*).	2d speed, or 50 F.P.M. Hand feed.	Forged 29° thread tool, or holder and cutter, inside calipers, rule.
Arrange lathe for 5 threads with thread stop reversed. Rough thread to $1\frac{1}{4}''$ the diameter of gage A, (4), fifteen cuts .006″ each, five cuts .002″ each. Depth of thread .100″.	1st speed, or 30 F.P.M.	Without oil.
Start tap in lathe, pull belt downward and follow with dead center. Be sure that tap follows thread or it may ream out thread. Remove nut and tap to reaming stand or vise, and finish tapping, (5).	Reaming stand, or vise and grooved wooden jaws.	$1\frac{1}{4}'' \times 5$, 29° thread tap and tap wrench. Lubricate tap freely with lard oil.

Digitized by Google

SCHEDULE OF OPERATIONS, MACHINES AND TOOLS.
Concluded.

Operations.	Machines, Speeds, Feeds.	Tools.
Mount nut on nut mandrel and rough square, (6), one or two cuts.	2d or 3d speed, or 40 F.P.M.	$1\frac{1}{4}'' \times 5$, 29° thread nut mandrel, dog, round-nose tool or holder and cutter, calipers, rule.
Finish square, (6), one or two cuts.	3d or 4th speed, or 60 F.P.M.	Facing tool, or holder and cutter.
Reverse nut and square to length to remove recess, (7), two or three cuts.	2d or 3d speed, or 60 F.P.M.	Round-nose tool, or holder and cutter and facing tool.
Rough turn, (8), one or two cuts.	1st or 2d speed, or 35 F.P.M. Medium power feed.	Diamond-point tool or holder and cutter, 15° rake, calipers, rule.
Finish turn, (8), one cut. File, (9).	2d or 3d speed, or 50 F.P.M. 4th speed, or 175 F.P.M.	Diamond-point or round-nose tool, or holder and cutter, calipers, rule. 8″ or 10″ mill bastard file.
Nurl, (10), two to four times. See Machine Nurling, § 370.	1st or 2d speed, or 35 F.P.M. Medium power feed — 80 to 1″.	Machine nurling tool, medium pitch, lard oil.
File corners slightly to remove burr, (11), (12).	4th speed, or 175 F.P.M.	8″ or 10″ mill bastard file.

Attention. — In the absence of a tap the inside 29° thread tool may be used to finish the thread but the thread will not be as smooth or the fit of screw and nut as good.

Note. — The nut in Fig. 415 is nurled for convenience in handling as a problem. For practical styles of nuts to transmit motion, see §626.

MULTIPLE THREADS.

622. Multiple-threaded screws, such as double and triple threads, etc., in Square, 29°, and other forms of threads, are used in cases where a quick lead is required, but a deep thread is not desirable.

Digitized by Google

623. To cut double Square thread, Fig. 416, 8 threads to 1″, pitch $\frac{1}{8}$″.

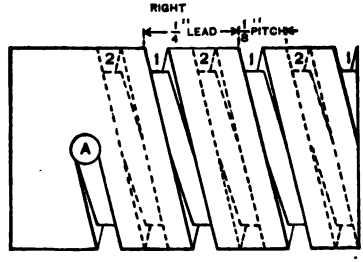


FIG. 416. — DOUBLE SQUARE THREAD SCREW.

SCHEDULE OF OPERATIONS.

1. Gear lathe to cut 4 threads per inch.
2. Drill hole *A* to terminate first groove.
3. Cut groove 1 to diameter.
4. If ratio of stud spindle and lathe spindle is 1 to 1, and stud gear has an even number of teeth, mark with chalk a tooth on stud gear and the corresponding space in idler gear. Then mark a tooth on stud gear diametrically opposite, which is determined by counting half the teeth in the stud gear.

   Disengage idler gear from stud gear, rotate lathe spindle and bring gears into mesh as indicated by chalk marks on the teeth of the gears.
5. Drill hole *A′* (not shown) opposite *A* to terminate second groove.
6. Cut groove 2 to diameter.

Attention.— To cut a triple thread, the lathe spindle is rotated one-third of a revolution to cut the second thread, and another third to cut the third thread. If gear on stud is not divisible by number of threads to be cut, select change gears that have a stud gear that is.

Special face plates with multiple equidistant slots (index milled) are convenient for cutting multiple threads, as the tail of the dog can be more readily shifted than the gears after each thread is cut.

624. Width and inclination of tool for multiple threads. — The width or shape of tools for multiple threads is governed by pitch of the screw; and the inclination by the lead of the screw.

625. Multiple-thread taps are similar to those for square and 29° thread nuts. See §§ 608, 619. The Sharp V and 29° forms of this thread can be cut with dies.

Digitized by Google

ALINEMENT DRILLING AND TAPPING.

626. Fixed nuts, Fig. 417, which receive screws to transmit motion are made in various forms, as bushing nut *A*, which is threaded and forced into a bored hole and used to

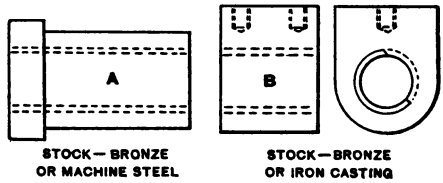


FIG. 417. — BUSHING AND BRACKET NUTS.

receive vertical feed screw on a milling machine, or bracket nut *B*, used to receive cross-slide screw on a lathe, or bracket nut *C*, Fig. 418, for cross-slide screw on a milling machine.

Bracket nuts are drilled and tapped in alinement with scraped slide and bearing of screw, and the work may be done with a regular jig, or with a part of the machine itself used as an improvised jig, as knee *B*, Fig. 418.

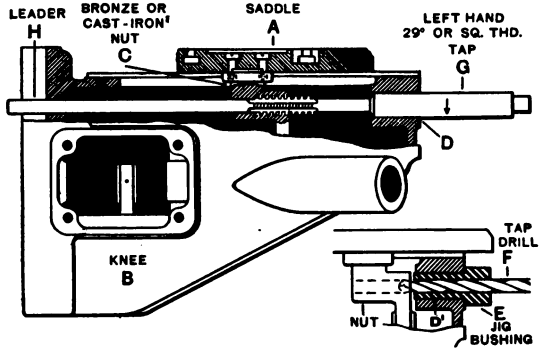


FIG. 418. — ALINEMENT DRILLING AND TAPPING, MILLING MACHINE CROSS-FEED SCREW NUT. HOLE IN KNEE USED AS A JIG.

627. To drill and tap cross-feed screw nut in axial alinement in milling machine saddle and knee, Fig. 418. **Improvised jig, 29° or Square thread tap.**

Digitized by Google

SCHEDULE OF OPERATIONS.

Place saddle *A* on knee *B*. Fit and bolt nut *C* to saddle and move close to hole *D*.

Clamp saddle and knee to angle plate at vertical drilling machine or to table of horizontal drilling machine and aline with spindle.

In hole *D'* insert jig bushing *E*, and drill nut with tap drill *F*.

Move saddle and nut away from hole *D*, and clamp saddle. Remove bushing, insert tap *G*, and tap hole by hand. Three or four taps of increasing diameters are used.

Attention. — Taps without leaders are used, but taps with leaders, either threaded or plain, as at *H*, are preferred for accuracy.

ECCENTRIC TURNING.

628. Eccentric turning. — Besides ordinary straight and aper turning, in which there is only a single axis and a single pair of centers, there is another class of turning, known as eccentric or offset turning, in which there is more than one axis and consequently more than one pair of centers.

629. Laying out and turning an eccentric shaft, Fig. 419, which has a throw of three-eighths of an inch.

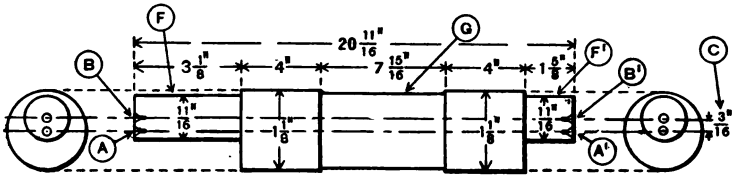


FIG. 419. — SCHEDULE DRAWING OF ECCENTRIC SHAFT FOR BACK GEARS.

SCHEDULE OF OPERATIONS.

1. Center stock as at *AA'* Fig. 419. Rough and finish square ends in regular way. Rough turn diameter.

2. Lay out eccentric centers *BB'*, distance at *C*, as follows: Mount shaft on centers, clamp pointed tool in tool post and make short lines at each end in same plane as at (1) and (2) Fig. 420. Draw radial lines on each end as at *D* with pointed tool or center square and scriber. Slant pointed tool to the left with point 3/16" (half the required throw of eccentric) from lathe center and draw

Digitized by Google

SCHEDULE OF OPERATIONS. *Concluded.*

lines at both ends intersecting radial lines as at *E*. Make center punch marks at these intersections, and drill and countersink in the regular way.

Mount in lathe on these eccentric centers *BB'*, Fig. 419, and measure throw with indicator or use paper between lathe tool and work. Scrape over countersinks to correct throw or to make throw uniform at both ends.

Rough turn and finish square reduced portions *FF'*. Change to regular centers *AA'*, and finish large diameter. Reduce *G* $\frac{1}{16}$″ in diameter for clearance.

Attention. — To save time in making a number of eccentric shafts, make a jig to drill and countersink ends.

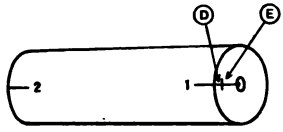


FIG. 420. — LOCATING SECOND PAIR OF CENTERS ON AN ECCENTRIC SHAFT.

630. To turn engine eccentric, Fig. 421.

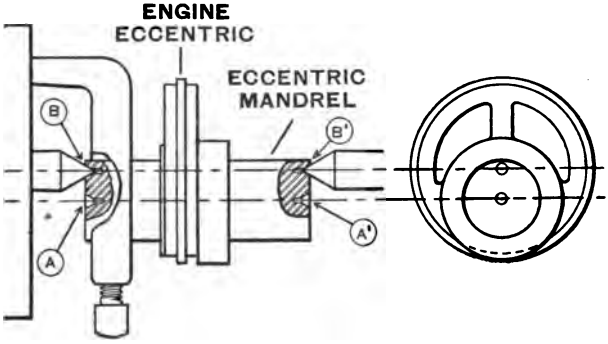


FIG. 421. — TURNING AN ENGINE ECCENTRIC.

SCHEDULE OF OPERATIONS.

1. Make eccentric mandrel Fig. 421, of required throw by method, § 629.

2. Chuck, ream, and push eccentric casting lightly onto mandrel. Mount in lathe and rap eccentric until it runs true. Take from lathe and press mandrel hard. On large eccentrics, set screws are used to fasten eccentric to mandrel. Do all facing operations and turning of

Digitized by Google

SCHEDULE OF OPERATIONS. *Concluded.*

hubs on regular centers AA', and all eccentric diameters on eccentric centers BB'. *Attention.* — When eccentric centers lie outside shaft, eccentric may be turned on disk, carrying	split stud with expanding screw, the whole being bolted to face plate of lathe in position to give desired throw. Some prefer this fixture to an eccentric mandrel.

631. Crankshaft turning. Fig. 422. — One of the most common forms of eccentric turning is an engine crankshaft. Here the centers or axes lie outside of the shaft, so that fixtures, arms or flanges, carrying the centers must be provided.

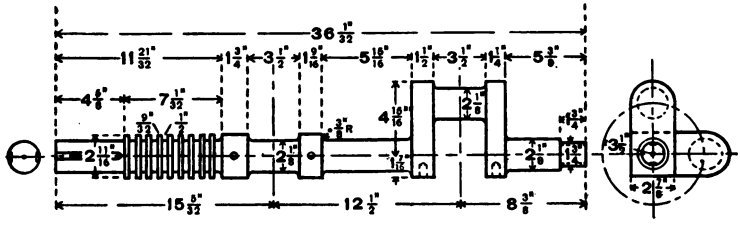


FIG. 422. — WORKING DRAWING OF TWO-THROW 90° ENGINE CRANKSHAFT.

Crankshafts are made with one or more cranks and may have solid eccentrics as well. Small crankshafts, in the rough, may be obtained drop-forged, of machine steel, cut from steel slabs or in steel castings. The larger sizes come partly machined after being forged under the steam hammer.

632. To lay out two-throw 90° crankshaft center fixtures, Fig. 423.

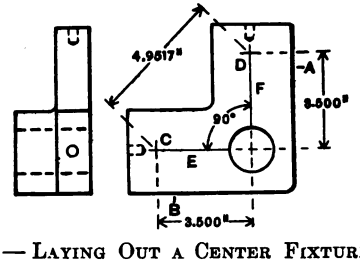


FIG. 423. — LAYING OUT A CENTER FIXTURE FOR A TWO-THROW 90° ENGINE CRANKSHAFT.

Digitized by Google

SCHEDULE OF OPERATIONS.

1. In two fixture castings chuck and ream holes to next regular diameter larger than required diameter of ends of crankshaft, mount on mandrel and square arms and hub. Plane edges *AB*, at right angles and an equal distance from hole.

2. Set pointed tool $3\frac{1}{2}''$ from point of dead center and describe arcs *CD*. If fixtures are not planed, points *CD* are obtained as follows:

Center punch point on *C*, in middle casting on the arc. Set dividers 4.9517″ the hypothenuse of right triangle whose base is $3\frac{1}{2}''$, using the one-hundredth graduation of the rule, and with *C* as center intersect arc at *D*. Distance *CD*, may be obtained by the following rule: Find the square root of the sum of the squares of the two throws.

Example: $\sqrt{3\frac{1}{2}^2 + 3\frac{1}{2}^2} = 4.9517$, answer. Lines *EF*, on each fixture are used to aline cranks with fixtures when sides of fixtures are not 90°.

Attention. — If only one crankshaft is to be turned, the fixture may be drilled and countersunk in casting. For a number of crankshafts, chuck large holes and drive in hardened and ground steel plugs carrying large center holes.

To make these holes in correct location, clamp fixtures to face plate of lathe with marks *C* and *D*, in axis of rotation as tested with axis indicator (see Lathe Axis Indicator, § 279), then drill and bore to size.

633. To aline center fixtures and lay out cranks, Fig. 424.

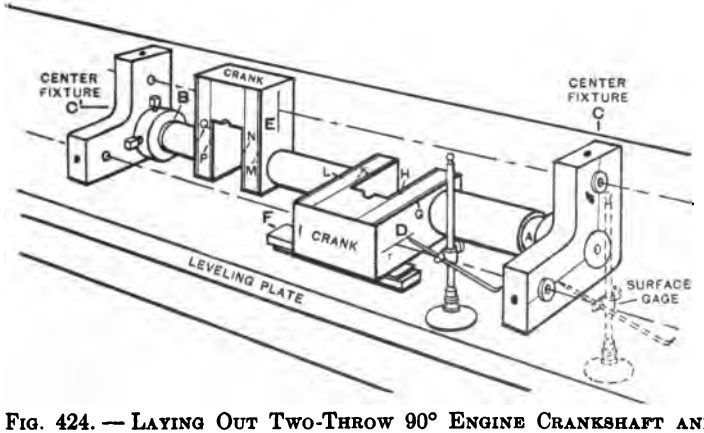


FIG. 424. — LAYING OUT TWO-THROW 90° ENGINE CRANKSHAFT AND ALINING CRANKS WITH CENTER FIXTURES.

Digitized by Google

SCHEDULE OF OPERATIONS.

To adjust fixtures, center and square shaft in relation to crank webs.

Turn *AB* to fit center fixtures *CC'*.

Mount fixtures and adjust centers of cranks *D* and *E* to aline with centers of fixtures by wedges *F* and test with surface gage. Lay out crank web by lines and center punch at *G*, *H*, *K*, *L*, *M*, *N*, *P*, *Q*.

Mount in lathe on one pair of crank centers and revolve; regular centers are horizontally opposite. Test top of turned ends with surface gage.

634. To turn crankshaft, Fig. 425.

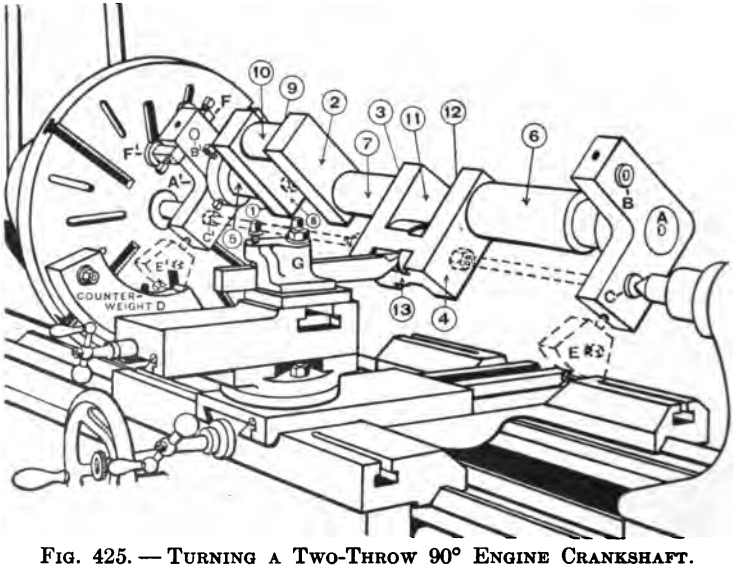


FIG. 425. — TURNING A TWO-THROW 90° ENGINE CRANKSHAFT. SCHEDULE DRAWING.

SCHEDULE OF OPERATIONS.

Rough turn blank. Mount upon regular centers *AA'*, Fig. 425. Rough square faces of webs 1, 2, 3, 4, and rough turn shaft 5, 6, and 7.

Change to crank centers *BB'*. Rough square and turn 8, 9, and 10.

Change to crank centers *CC'* and rough square and turn 11, 12, 13.

Counterbalance crank fixtures by using weight *D*, or preferably by adjustable weights *EE'*.

Use driver *FF'* with piece of leather to reduce jar.

Digitized by Google

SCHEDULE OF OPERATIONS. *Concluded.*

Season crankshaft between roughing and finishing, if time will permit. *Finish square and turn in reverse order.* It is best to spot 7 and use	steady rest for finishing. For slender crankshafts use jack or braces as shown by dotted lines. European tool post *G* is preferred to a single tool post. Mill or plane edges of webs.

Attention. — Crankshaft lathes are obtainable. Automobile and motorboat crankshafts may be rough turned, then ground, or ground direct from drop forgings or steel castings. See *Advanced Machine Work.*

Digitized by Google

CHAPTER XIX.

TABLES AND OTHER DATA USED IN MACHINE WORK.

635. Electrical units.

VOLT. — The unit of electro-motive force, (E.M.F.) The force required to send one ampere of current through one ohm of resistance.

AMPERE. — The unit of current. The current which will pass through one ohm resistance when impelled by one volt. A milli-ampere = one-thousandth of an ampere.

OHM. — The unit of resistance. The resistance offered to the passage of one ampere when impelled by one volt. The megohm = one million ohms.

WATT. — The unit of power:

One ampere × one volt = one watt.
Amperes × volts = watts.
$(\text{Amperes})^2 \times \text{ohms} = \text{watts}$.
$(\text{Volts})^2 \div \text{ohms} = \text{watts}$.
746 watts = 1 horse power.
1000 watts = 1 kilo watt,
approximately $1\frac{1}{3}$ horse power.

COULOMB. — The quantity of current which impelled by one volt would pass through one ohm in one second.

JOULE. — The unit of work. The work done by one watt in one second.

Digitized by Google

636. International and French Standard threads.

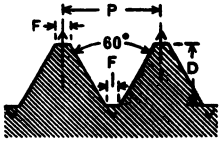


FIG. 426.

Diameter and pitch in Metric Measure.

Formula $\begin{cases} p = \text{pitch.} \\ d = \text{depth} = p \times .64952. \\ f = \text{flat} = \frac{p}{8}. \end{cases}$

637. International Standard thread.

Diameter in Millimeters.	Pitch in Millimeters.	Diam. at Root of Thread, m/m.	Diameter in Millimeters.	Pitch in Millimeters.	Diam. at Root of Thread, m/m.	Diameter in Millimeters.	Pitch in Millimeters.	Diam. at Root of Thread, m/m.
6	1.0	4.70	20	2.5	16.75	48	5.0	41.51
7	1.0	5.70	22	2.5	18.75	52	5.0	45.51
8	1.25	6.38	24	3.0	20.10	56	5.5	48.86
9	1.25	7.58	27	3.0	23.10	60	5.5	52.86
10	1.5	8.05	30	3.5	25.45	64	6.0	56.21
11	1.5	9.05	33	3.5	28.45	68	6.0	60.21
12	1.75	9.73	36	4.0	30.80	72	6.5	63.56
14	2.0	11.40	39	4.0	33.80	76	6.5	67.56
16	2.0	13.40	42	4.5	36.15	80	7.0	70.91
18	2.5	14.75	45	4.5	39.15	.....	.....	.....

638. French Standard thread.

Diameter in Millimeters.	Pitch in Millimeters.	Diam. at Root of Thread, m/m.	Diameter in Millimeters.	Pitch in Millimeters.	Diam. at Root of Thread, m/m.	Diameter in Millimeters.	Pitch in Millimeters.	Diam. at Root of Thread, m/m.
3	0.5	2.35	16	2.0	13.40	36	4.0	30.80
4	0.75	3.03	18	2.5	14.75	38	4.0	32.80
5	0.75	4.03	20	2.5	16.75	40	4.0	34.80
6	1.0	4.70	22	2.5	18.75	42	4.5	36.15
7	1.0	5.70	24	3.0	20.10	44	4.5	38.15
8	1.0	6.70	26	3.0	22.10	46	4.5	40.15
9	1.0	7.70	28	3.0	24.10	48	5.0	41.51
10	1.5	8.05	30	3.5	25.45	50	5.0	43.51
12	1.5	10.05	32	3.5	27.45	.....	.....	.....
14	2.0	11.40	34	3.5	29.45	.....	.....	.....

Digitized by Google

639. The American licensed automobile manufacturers' standard screws and nuts.

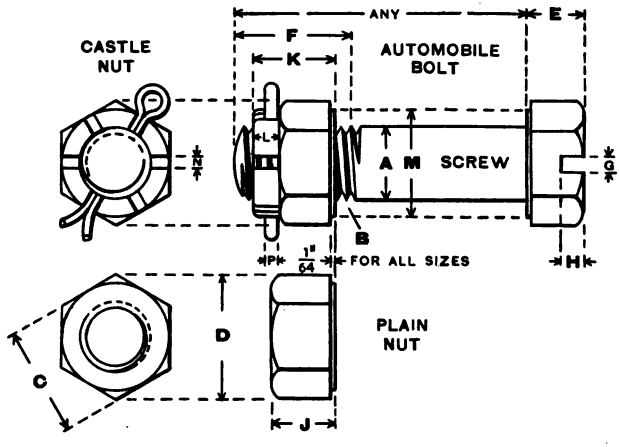


FIG. 427.

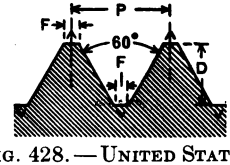


FIG. 428. — UNITED STATES STANDARD THREAD.

Formula $\begin{cases} p = \text{pitch.} \\ d = \text{depth} = p \times 64{,}952. \\ f = \text{flat} = \frac{p}{8}. \end{cases}$

For sizes of tap drills, see §§ 537, 640.

Diameter of screw = nominal diameter − .001″.

Thread = U. S. S. in form but with finer pitches. Taps and dies are marked U.S.F.

Heads and nuts are semi-finished but smaller than U. S. S.

Screws soft. Plain nuts soft. Castle nuts case-hardened. Nuts should be a good fit on screw— without perceptible shake.

The tap is from .002″ to .003″ larger than standard at the top of thread to give the screw clearance in the nut. Material for screws and nuts, machine steel; tensile strength, 100,000 lbs. per square inch; elastic limit, 60,000 lbs. per square inch.

Threaded portions of screw should be one and one-half times the body diameter.

Attention. — The castle nut is used where a positive locking system is desired.

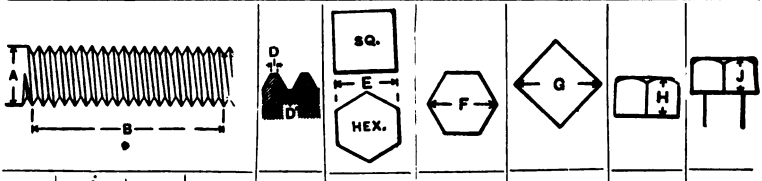
Digitized by Google

640. Table of American Licensed Automobile Manufacturers' Standard Screws and Nuts.

Nominal Diameter of Screw.	A	Threads per Inch.	B	Sizes of Tap Drills.	Sizes of Body Drills.	Across Flats of Head and Nut.	C	Across Corners of Head and Nut.	D	Thickness of Head.	E	Length of Thread.	F	Width of Slot in Head.	G	Depth of Slot in Head.	H	Thickness of Plain Nut.	J	Thickness of Castle Nuts.	K	Height of Castle, also Depth of Slots.	L	Diameter of Facing under Head and Nut, also Diameter of Castle.	M	Width of Slot in Castle and Diameter of Cotter Pin Hole.	N	Diameter of Cotter Pin.	P
	1/4		28		17/64		7/16		.043		3/16		3/8		1/16		3/32		7/32		9/32		3/32		1/4		5/64		1/16
	5/16		24		21/64		7/16		.057		3/16		3/8		1/16		3/32		7/32		9/32		3/32		1/4		5/64		1/16
	3/8		24	No. 5	25/64		1/2		.064		7/32		7/16		1/16		3/32		1/4		5/16		3/32		5/16		3/32		3/32
	7/16		20	G	29/64		9/16		.079		1/4		1/2		1/16		3/32		1/4		5/16		3/32		5/16		3/32		3/32
	1/2		20	P	33/64		5/8		.086		9/32		1/2		1/16		3/32		1/4		5/16		3/32		5/16		3/32		3/32
	9/16		18	3/8	37/64		11/16		1.010		5/16		9/16		1/16		3/32		1/4		5/16		3/32		5/16		3/32		3/32
	5/8		18	I/2	17/32		3/4		1.082		11/32		9/16		1/16		3/32		1/4		5/16		3/32		5/16		3/32		3/32
	11/16		16	9/16	19/32		13/16		1.155		3/8		5/8		1/16		3/32		1/4		5/16		3/32		5/16		3/32		3/32
	3/4		16	5/8	21/32		7/8		1.299		13/32		5/8		1/16		3/32		1/4		5/16		3/32		5/16		3/32		3/32
	1		14	11/16	23/32		1		1.443		7/16		11/16		1/16		3/32		1/4		5/16		3/32		5/16		3/32		3/32
			14	3/4	25/32		1 1/16		1.660		15/32		11/16		1/16		3/32		1/4		5/16		3/32		5/16		3/32		3/32

Digitized by Google

641. Table of United States Standard bolt heads and nuts.



Diam. of Screw. A	Diam. in Decimals.	Threads per Inch. B	Root Diam. of Screw. C	Width of Flats. D	Across Flats of Sq. and Hex. Nuts or Heads. E	Across Corners of Sq. and Hex. Nuts or Heads. F	Across Corners of Sq. Nut or Head. G	Thickness of Nut. H	Thickness of Head. J
$\frac{1}{4}$	.25	20	.185	.0056	$\frac{1}{2}$	$\frac{17}{64}$	.61	$\frac{1}{4}$	$\frac{1}{4}$
$\frac{5}{16}$	.3125	18	.2403	.0069	$\frac{19}{32}$	$\frac{11}{16}$	.75	$\frac{5}{16}$	$\frac{19}{64}$
$\frac{3}{8}$	.375	16	.2996	.0078	$\frac{11}{16}$	$\frac{51}{64}$	.88	$\frac{3}{8}$	$\frac{27}{64}$
$\frac{7}{16}$	.4375	14	.3447	.0089	$\frac{25}{32}$	$\frac{9}{10}$	1.01	$\frac{7}{16}$	$\frac{31}{64}$
$\frac{1}{2}$	.5	13	.4001	.0096	$\frac{7}{8}$	1	1.14	$\frac{1}{2}$	$\frac{7}{16}$
$\frac{9}{16}$	.5625	12	.4542	.0104	$\frac{31}{32}$	$1\frac{1}{8}$	1.28	$\frac{9}{16}$	$\frac{35}{64}$
$\frac{5}{8}$	.625	11	.5069	.0114	$1\frac{1}{16}$	$1\frac{7}{32}$	1.41	$\frac{5}{8}$	$\frac{7}{16}$
$\frac{3}{4}$	.75	10	.6201	.0125	$1\frac{1}{4}$	$1\frac{7}{16}$	1.67	$\frac{3}{4}$	$\frac{43}{64}$
$\frac{7}{8}$	.875	9	.7307	.0139	$1\frac{7}{16}$	$1\frac{21}{32}$	1.94	$\frac{7}{8}$	$\frac{25}{32}$
1	.....	8	.8376	.0156	$1\frac{5}{8}$	$1\frac{7}{8}$	2.20	1	$\frac{29}{32}$
$1\frac{1}{8}$	1.	7	.9394	.0179	$1\frac{13}{16}$	$2\frac{3}{32}$	2.47	$1\frac{1}{8}$	$\frac{33}{64}$
$1\frac{1}{4}$	1.125	7	1.0644	.0179	2	$2\frac{5}{16}$	2.74	$1\frac{1}{4}$	1
$1\frac{3}{8}$	1.25	6	1.1585	.0208	$2\frac{3}{16}$	$2\frac{17}{32}$	3.00	$1\frac{3}{8}$	$1\frac{3}{32}$
$1\frac{1}{2}$	1.375	6	1.2835	.0208	$2\frac{3}{8}$	$2\frac{3}{4}$	3.27	$1\frac{1}{2}$	$1\frac{3}{16}$
$1\frac{5}{8}$	1.5	$5\frac{1}{2}$	1.3888	.0227	$2\frac{9}{16}$	$2\frac{31}{32}$	3.53	$1\frac{5}{8}$	$1\frac{37}{64}$
$1\frac{3}{4}$	1.625	5	1.4902	.0250	$2\frac{3}{4}$	$3\frac{3}{16}$	3.70	$1\frac{3}{4}$	$1\frac{5}{8}$
$1\frac{7}{8}$	1.75	5	1.6152	.0250	$2\frac{15}{16}$	$3\frac{3}{32}$	4.06	$1\frac{7}{8}$	$1\frac{45}{64}$
2	1.875	$4\frac{1}{2}$	1.7113	.0278	$3\frac{1}{8}$	$3\frac{5}{8}$	4.33	2	$1\frac{9}{16}$
$2\frac{1}{4}$	2.25	$4\frac{1}{2}$	1.9613	.0278	$3\frac{1}{2}$	$4\frac{1}{16}$	5.39	$2\frac{1}{4}$	$1\frac{3}{4}$
$2\frac{1}{2}$	2.5	4	2.1752	.0313	$3\frac{7}{8}$	$4\frac{1}{2}$	5.93	$2\frac{1}{2}$	$1\frac{15}{16}$
$2\frac{3}{4}$	2.75	4	2.4252	.0313	$4\frac{1}{4}$	$4\frac{29}{32}$	6.45	$2\frac{3}{4}$	$2\frac{1}{8}$
3	3.25	$3\frac{1}{2}$	2.6288	.0357	$4\frac{5}{8}$	$5\frac{3}{8}$	6.98	3	$2\frac{5}{16}$

Attention. — A bolt is usually threaded a distance equal to twice the body diameter.

Digitized by Google

642. Formulas of bolt heads and nuts. — While finished heads and nuts (U. S. S.) are often made $\frac{1}{16}''$ smaller than the rough, it is best to make both the *same* size and to use the *same* wrench.

The short diameter or width across flats = $\frac{1}{2}$ × (diameter of bolt) + $\frac{1}{8}''$.

The long diameter or distance across corners of square head or nut = short diameter × 1.414.

The long diameter of hexagonal head or nut = short diameter × 1.155.

Thickness of nut = diameter of bolt.

Thickness of head = $\frac{1}{2}$ short diameter of head.

643. Weights of castings from wooden patterns.

Materials of Patterns	Iron.	Composition.	Yellow Brass.	Copper.	Bronze.	Aluminium.	Steel.	Lead.	Zinc.
Pine	16½	18½	18	18	18	6	16	26	16½
Mahogany	14¾	15	14	14	14	5¼	14	23	14¾

644. Estimating weight of cored castings.

One cubic inch of metal weighs

Metal.	Lb.
Iron	.2627
Composition	.3023
Yellow brass	.2997
Copper	.3135
Bronze	.306
Aluminium	.0926
Steel	.281
Lead	.415
Zinc	.26

645. Shrinkage of castings.

Metal.	In. per Foot.
Iron	$\frac{1}{8}$
Composition	$\frac{3}{16}$
Yellow brass	$\frac{3}{16}$
Copper	$\frac{3}{16}$
Bronze or gun metal	$\frac{3}{16}$
Aluminium	$\frac{7}{32}$
Steel	$\frac{1}{4}$
Lead	$\frac{5}{16}$
Zinc	$\frac{5}{16}$

646. Shrink rules, that is special rules with graduations wider than standard measurement, are used for measuring when making patterns so that the pattern will be large enough to allow for contraction of casting in cooling and produce the proper size casting. These rules are obtainable with different allowances of shrinkage for different metals, as $\frac{1}{16}''$, $\frac{1}{12}''$, $\frac{1}{10}''$, $\frac{3}{32}''$, $\frac{1}{8}''$, $\frac{3}{16}''$, and $\frac{1}{4}''$ per foot.

Digitized by Google

647. Consecutive and comparative tables of different drill sizes.

ENGLISH, METRIC, NUMBER, LETTER.

INCH.	M.M.	WIRE GAGE.	DECIMALS OF AN INCH	INCH.	M.M.	WIRE GAGE.	DECIMALS OF AN INCH	INCH.	M.M.	WIRE GAGE.	DECIMALS OF AN INCH
....	.....	80	.0135	....	.....	56	.0465	....	.....	40	.098
....	.....	79	.0145	3/64	.....	.....	.046875	....	2.5	.....	.098425
1/64	.....	.....	.015625	....	1.2	.....	.047244	....	.....	39	.0995
....	.4	.....	.01574	....	1.3	.....	.051181	....	.....	38	.1015
....	.....	78	.016	....	.....	55	.052	....	2.6	.....	.102362
....	.....	77	.018	....	.....	54	.055	....	.....	37	.104
....	.5	.....	.01968	....	1.4	.....	.055118	....	2.7	.....	.1063
....	.....	76	.020	....	1.5	.....	.05905	....	.....	36	.1065
....	.....	75	.021	....	.....	53	.0595	7/64	.....	.....	.109375
....	.....	74	.0225	1/16	.....	.....	.0625	....	.....	35	.11
....	.6	.....	.02362	....	1.6	.....	.06299	....	2.8	....	.11024
....	.....	73	.024	....	.....	52	.0635	....	.....	34	.111
....	.....	72	.025	....	1.7	.....	.066929	....	.....	33	.113
....	.....	71	.026	....	.....	51	.067	....	2.9	.....	.11417
....	.7	.....	.02756	....	.....	50	.07	....	.....	32	.116
....	.....	70	.028	....	1.8	.....	.070866	....	3	.....	.11811
....	.....	69	.02925	....	.....	49	.073	....	....	31	.12
....	.....	68	.031	....	1.9	.....	.0748	....	3.1	.....	.12205
1/32	.....	.....	.03125	....	.....	48	.076	1/8	.....	.....	.125
....	.8	.....	.031496	5/64	.....	.....	.078125	....	3.2	.....	.12598
....	.....	67	.032	....	.....	47	.0785	....	.....	30	.1285
....	.....	66	.033	....	2	.....	.07874	....	3.3	.....	.12992
....	.....	65	.035	....	.....	46	.081	....	3.4	.....	.13386
....	.9	.....	.03543	....	.....	45	.082	....	.....	29	.136
....	.....	64	.036	....	2.1	.....	.082677	....	3.5	.....	.1378
....	.....	63	.037	....	.....	44	.086	....	.....	28	.1405
....	.....	62	.038	....	2.2	.....	.086614	9/64	.....	.....	.140625
....	.....	61	.039	....	.....	43	.089	....	3.6	.....	.14173
....	1	.....	.03937	....	2.3	.....	.09055	....	.....	27	.144
....	.....	60	.04	....	.....	42	.0935	....	3.7	.....	.14567
....	.....	59	.041	3/32	.....	.....	.09375	....	.....	26	.147
....	.....	58	.042	....	2.4	.....	.09448	....	.....	25	.1495
....	.....	57	.043	....	.....	41	.096..	....	3.8	.....	.14961
....	1.1	.....	.043307	....	.....	.....	.....	....	.....	.....	.....

Digitized by Google

Consecutive and comparative tables of different drill sizes. *Continued.*

ENGLISH, METRIC, NUMBER, LETTER.

INCH.	M.M.	WIRE GAGE.	DECIMALS OF AN INCH	INCH.	M.M.	WIRE GAGE.	DECIMALS OF AN INCH	INCH.	M.M.	WIRE GAGE.	DECIMALS OF AN INCH
....	.....	24	.152	....	4.5	.....	.17717	13/64	.....	.....	.203125
...	3.9	.....	.15354	....	.....	15	.18	....	.....	6	.204
....	.....	23	.154	....	4.6	.....	.1811	....	5.2	.....	.20473
5/32	.....	.....	.15625	....	.....	14	.182	....	.....	5	.2055
....	.....	22	.157	....	.....	13	.185	....	5.3	.....	.20866
....	4	.....	.15748	....	4.7	.....	.18504	....	.....	4	.209
....	.....	21	.159	3/16	.....	.....	.1875	....	5.4	.....	.2126
....	.....	20	.161	....	4.8	.....	.18898	....	.....	3	.213
....	4.1	.....	.16142	....	.....	12	.189	....	5.5	.....	.21654
....	4.2	.....	.16536	....	.....	11	.191	7/32	.....	.....	.21875
....	.....	19	.166	....	4.9	.....	.19291	....	5.6	.....	.22047
....	4.3	.....	.16929	....	.....	10	.1935	....	.....	2	.221
....	.....	18	.1695	....	.....	9	.196	....	5.7	.....	.22441
11/64	.....	.....	.171875	....	5	.....	.19685	....	.....	1	.228
....	.....	17	.173	....	.....	8	.199	....	5.8	.....	.22835
....	4.4	.....	.17323	....	5.1	.....	.20079	....	5.9	.....	.23228
....	.....	16	.177	....	.....	7	.201	....	.....	.....	.....

INCH.	M.M.	LETTER SIZES.	DECIMALS OF AN INCH	INCH.	M.M.	LETTER SIZES.	DECIMALS OF AN INCH	INCH.	M.M.	LETTER SIZES.	DECIMALS OF AN INCH
....	.....	A	.234	....	.....	H	.266	....	.....	N	.302
15/64	.....	.....	.234375	....	6.8	.....	.26772	....	7.7	.....	.30314
....	6	.....	.23622	....	6.9	.....	.27165	....	7.8	.....	.30709
....	.....	B	.238	....	.....	I	.272	....	7.9	.....	.31102
....	6.1	.....	.24015	....	7	.....	.27559	5/16	.....	.....	.3125
....	.....	C	.242	....	.....	J	.277	....	8	.....	.31496
....	6.2	.....	.2441	....	7.1	.....	.27952	....	.....	O	.316
....	.....	D	.246	....	.....	K	.281	....	8.2	.....	.32284
....	6.3	.....	.24803	9/32	.....	.....	.28125	....	.....	P	.323
1/4	.....	E	.25	....	7.2	.....	.28347	21/64	.....	.....	.328125
....	6.4	.....	.25197	....	7.3	.....	.2874	....	.....	Q	.332
....	6.5	.....	.25591	....	.....	L	.29	....	8.5	.....	.33465
....	.....	F	.257	....	7.4	.....	.29133	....	8.6	.....	.33859
....	6.6	.....	.25984	....	.....	M	.295	....	.....	R	.339
....	.....	G	.261	....	7.5	.....	.29528	11/32	.....	.....	.34375
....	6.7	.....	.26377	19/64	.....	.....	.296875	....	8.8	.....	.34646
17/64	.....	.....	.265625	....	7.6	.....	.29922	....	.....	S	.348

Digitized by Google

Consecutive and comparative tables of different drill sizes. *Continued.*

ENGLISH, METRIC, NUMBER, LETTER.

IN.	M.M.	LETTER SIZES.	DECIMALS OF AN INCH.	IN.	M.M.	DECIMALS OF AN INCH.	IN.	M.M.	DECIMALS OF AN INCH.
....	9	.....	.35433	9/16	.....	.5625	55/64	.....	.859375
....	.....	T	.358	....	14.5	.57087	....	22	.86614
23/64	.....	.....	.359375	37/64	.....	.578125	7/8	.....	.875
....	9.2	.....	.36221	....	15	.59055	....	22.5	.88583
....	.....	U	.368	19/32	.....	.59375	57/64	.....	.890625
....	9.5	.....	.37402	39/64	.....	.609375	....	23	.90551
3/8	.....	.....	.375	....	15.5	.61024	29/32	.....	.90625
....	.....	V	.377	5/8	.....	.625	59/64	.....	.921875
....	9.6	.....	.37796	....	16	.62992	....	23.5	.9252
....	9.8	.....	.38583	41/64	.....	.640625	15/16	.....	.9375
....	.....	W	.386	....	16.5	.6496	....	24	.94488
25/64	.....	.....	.390625	21/32	.....	.65625	61/64	.....	.953125
....	10	.....	.3937	....	17	.66929	....	24.5	.9646
....	.....	X	.397	43/64	.....	.671875	31/32	.....	.96875
....	.....	Y	.404	11/16	.....	.6875	....	25	.98425
13/32	.....	.....	.40625	....	17.5	.689	63/64	.....	.984375
....	.....	Z	.413	45/64	.....	.703125	1	.....	1.
....	10.5	.....	.4134	....	18	.70866	....	25.5	1.004
27/64	.....	.....	.421875	23/32	.....	.71875	1 1/64	.....	1.015625
....	11	.....	.43307	....	18.5	.72835	....	26	1.02362
7/16	.....	.....	.4375	47/64	.....	.734375	1 1/32	.....	1.03125
....	11.5	.....	.45276	....	19	.74803	....	26.5	1.0433
29/64	.....	.....	.453125	3/4	.....	.75	1 3/64	.....	1.046875
15/32	.....	.....	.46875	49/64	.....	.765625	1 1/16	.....	1.0625
....	12	.....	.47244	....	19.5	.76772	....	27	1.063
31/64	.....	.....	.484375	25/32	.....	.78125	1 5/64	.....	1.078125
....	12.5	.....	.4921	....	20	.7874	....	27.5	1.08268
1/2	.....	.....	.5	51/64	.....	.796875	1 3/32	.....	1.09375
....	13	.....	.51181	....	20.5	.8071	....	28	1.1024
33/64	.....	.....	.515625	13/16	.....	.8125	1 7/64	.....	1.109375
17/32	.....	.....	.53125	....	21	.82677	....	28.5	1.122
....	13.5	.....	.5315	53/64	.....	.828125	1 1/8	.....	1.125
35/64	.....	.....	.546875	27/32	.....	.84375	1 9/64	.....	1.140625
....	14	.....	.55118	....	21.5	.84646	....	29	1.1417

Digitized by Google

Consecutive and comparative tables of different drill sizes.

Concluded.

ENGLISH, METRIC, NUMBER, LETTER.

IN.	M.M.	DECIMALS OF AN INCH.	IN.	M.M.	DECIMALS OF AN INCH.	IN.	M.M.	DECIMALS OF AN INCH.
$1\frac{5}{32}$	.....	1.15625	$1\frac{7}{16}$	.....	1.4375	.....	44	1.7323
.....	29.5	1.1614	$1\frac{29}{64}$	.....	1.45312	$1\frac{47}{64}$	.....	1.73437
$1\frac{11}{64}$	.....	1.171875	.....	37	1.4567	$1\frac{3}{4}$	.....	1.75
.....	30	1.1811	$1\frac{15}{32}$	.....	1.46875	.....	44.5	1.7519
$1\frac{3}{16}$	.....	1.1875	.....	37.5	1.4764	$1\frac{49}{64}$	.....	1.76562
.....	30.5	1.2008	$1\frac{31}{64}$	.....	1.48437	.....	45	1.7717
$1\frac{13}{64}$	.....	1.203125	.....	38	1.4961	$1\frac{25}{32}$	.....	1.78125
$1\frac{7}{32}$	.....	1.21875	$1\frac{1}{2}$	.....	1.5	.....	45.5	1.79138
.....	31	1.2205	$1\frac{33}{64}$	.....	1.51562	$1\frac{51}{64}$	.....	1.79687
$1\frac{15}{64}$	.....	1.234375	.....	38.5	1.51576	.....	46	1.811
.....	31.5	1.24016	$1\frac{17}{32}$	.....	1.53125	$1\frac{13}{16}$	.....	1.8125
$1\frac{1}{4}$	.....	1.25	.....	39	1.5354	$1\frac{53}{64}$	.....	1.82812
.....	32	1.2598	$1\frac{35}{64}$	.....	1.54687	.....	46.5	1.83
$1\frac{17}{64}$	.....	1.26562	.....	39.5	1.5551	$1\frac{27}{32}$	.....	1.84375
.....	32.5	1.2795	$1\frac{9}{16}$	.....	1.5625	.....	47	1.85047
$1\frac{9}{32}$	.....	1.28125	.....	40	1.5748	$1\frac{55}{64}$	.....	1.85937
$1\frac{19}{64}$	.....	1.29687	$1\frac{37}{64}$	.....	1.57812	.....	47.5	1.87016
.....	33	1.2992	$1\frac{19}{32}$	.....	1.59375	$1\frac{7}{8}$	.....	1.875
$1\frac{5}{16}$	.....	1.3125	.....	40.5	1.5945	.....	48	1.88985
.....	33.5	1.319	$1\frac{39}{64}$	.....	1.60937	$1\frac{57}{64}$	.....	1.89062
$1\frac{21}{64}$	.....	1.328125	.....	41	1.6142	$1\frac{29}{32}$	.....	1.90625
.....	34	1.3386	$1\frac{5}{8}$	.....	1.625	.....	48.5	1.90945
$1\frac{11}{32}$	.....	1.34375	.....	41.5	1.6338	$1\frac{59}{64}$	.....	1.92187
.....	34.5	1.3583	$1\frac{41}{64}$	.....	1.64062	.....	49	1.92913
$1\frac{23}{64}$	.....	1.359375	.....	42	1.6536	$1\frac{15}{16}$	.....	1.9375
$1\frac{3}{8}$	.....	1.375	$1\frac{21}{32}$	.....	1.65625	.....	49.5	1.9488
.....	35	1.378	$1\frac{43}{64}$	.....	1.67187	$1\frac{61}{64}$	.....	1.95312
$1\frac{25}{64}$	.....	1.39062	.....	42.5	1.6733	.....	50	1.9685
.....	35.5	1.3977	$1\frac{11}{16}$	.....	1.6875	$1\frac{31}{32}$	.....	1.96875
$1\frac{13}{32}$	.....	1.40625	.....	43	1.6929	$1\frac{63}{64}$	.....	1.98437
.....	36	1.4173	$1\frac{45}{64}$	.....	1.70312	.....	50.5	1.9882
$1\frac{27}{64}$	.....	1.421875	.....	43.5	1.71259	2	.....	2.
.....	36.5	1.437	$1\frac{23}{32}$	.....	1.71875	.....	51	2.0079

Digitized by Google

648. Morse tapers with diagram and table of proportional parts.

For holes in drilling machine spindles, lathe spindles, shanks of drills, sockets, end mills, etc.

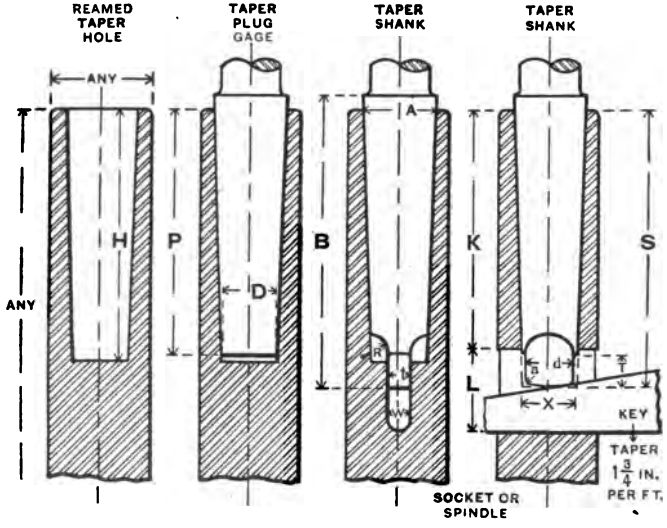


FIG. 429. — DIAGRAM OF MORSE TAPERS.

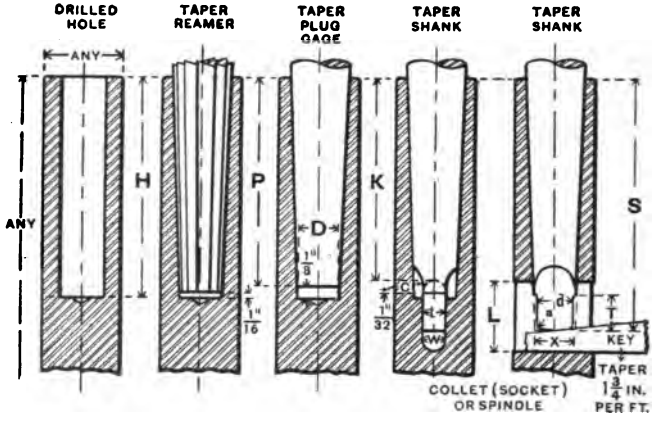
649. Table of Morse tapers.

Number of Taper	D	A	Shank. B	Shank. S	H	P	Tongue. t	Tongue. T	Tongue. R	Tongue. d	Tongue. A	Keyway. W	Keyway. L	K	X			
	Diam. of Plug at Small End.	Diam. at End of Socket.	Whole Length.	Depth.	Depth of Hole.	Standard Plug Depth.	Thick-ness.	Length.	Rad. of Mill.	Diam.	Radius.	Width.	Length.	End of Socket to Keyway.	Taper per Foot.	Taper per Inch.	Diam. of Point of Shank.	Limit for Tongue to Project through Gage.	No. of Key.	Rea. Projects through Gage.
0	.252	.3561	$2\frac{1}{16}$	$2\frac{7}{32}$	$2\frac{7}{32}$	2	$\frac{5}{32}$	$\frac{1}{4}$	$\frac{5}{32}$	.235	.04	.160	$\frac{9}{16}$	$1\frac{15}{16}$	.625	.05208	.2406	.003	0	$\frac{1}{32}$
1	.369	.475	$2\frac{9}{16}$	$2\frac{15}{32}$	$2\frac{1}{4}$	$2\frac{1}{16}$	$\frac{13}{64}$	$\frac{5}{16}$	$\frac{3}{16}$	.343	.05	.213	$\frac{3}{4}$	$2\frac{1}{16}$	.600	.05	.353	.003	1	$\frac{1}{32}$
2	.572	.700	$3\frac{1}{16}$	$2\frac{15}{16}$	$2\frac{5}{8}$	$2\frac{9}{16}$	$\frac{7}{32}$	$\frac{7}{16}$	$\frac{1}{4}$	$\frac{17}{64}$	.06	.260	$\frac{7}{8}$	$2\frac{1}{4}$	.602	.05016	.5527	.004	2	$\frac{3}{64}$
3	.778	.938	$3\frac{7}{16}$	$3\frac{11}{16}$	$3\frac{1}{4}$	$3\frac{1}{16}$	$\frac{1}{4}$	$\frac{9}{16}$	$\frac{5}{16}$	$\frac{33}{64}$	.08	.322	$1\frac{1}{16}$	$3\frac{1}{16}$	.602	.05016	.753	.004	3	$\frac{1}{16}$
4	1.020	1.231	$4\frac{7}{16}$	$4\frac{5}{8}$	$4\frac{1}{8}$	$4\frac{1}{16}$	$\frac{5}{16}$	$\frac{5}{8}$	$\frac{3}{8}$	$\frac{31}{32}$	.10	.478	$1\frac{1}{4}$	$3\frac{7}{8}$	.623	.05191	.991	.004	4	$\frac{1}{16}$
5	1.475	1.748	$6\frac{1}{4}$	$5\frac{7}{8}$	$5\frac{1}{4}$	$5\frac{3}{16}$	$\frac{3}{8}$	$\frac{3}{4}$	$\frac{7}{16}$	$1\frac{23}{32}$	.12	.635	$1\frac{1}{4}$	$4\frac{1}{16}$	.630	.0525	1.4395	.005	5	$\frac{1}{64}$
6	2.116	2.494	$8\frac{9}{16}$	$8\frac{1}{4}$	$7\frac{3}{8}$	$7\frac{1}{4}$	$\frac{7}{16}$	$1\frac{1}{16}$	$\frac{7}{16}$	2	.15	.760	$1\frac{1}{4}$	7	.626	.05216	2.0636	.005	6	$\frac{1}{64}$
7	2.750	3.270	$11\frac{5}{8}$	$11\frac{1}{4}$	$10\frac{1}{4}$	10	$\frac{1}{2}$	$1\frac{3}{8}$	$\frac{1}{2}$	$2\frac{5}{8}$	.18	1.135	$2\frac{1}{8}$	$9\frac{1}{4}$	.625	.05208	2.685	.005	7	$\frac{2}{64}$

Digitized by Google

650. Brown & Sharpe tapers with diagram and table of proportional parts.

For holes in milling machine spindles, collets, and shanks of arbors, end mills, etc.



TAPER $\frac{1}{2}$ IN. TO 1 FOOT EXCEPT NO. 10 WHICH IS .5161 PER FOOT

FIG. 430. — DIAGRAM OF BROWN & SHARPE TAPERS.

Digitized by Google

651. Table of Brown & Sharpe tapers.

No. of Taper.	Diam. of Plug at Small End. D	Plug Depth. P	Depth of Hole. H	Keyway from End of Spindle. K	Shank Depth. S	Length of Keyway. L	Width of Keyway. W	Length of Arbor Tongue. T	Diameter of Arbor Tongue. d	Thickness of Arbor Tongue. t	Radius of Tongue Circle. C	Radius of Tongue at "A." A	Diam. of Arbor at Small End Before it is Tongued. X	Limit for Tongue to Project through Test Tool Gage.
1	.20	15/16	1 1/16	15/16	1 3/16	3/8	.135	3/16	.170	1/8	3/16	.030	.189	.003
2	.25	1 3/16	1 5/16	1 11/64	1 1/2	1/2	.166	1/4	.220	5/32	3/16	.030	.237	.003
3	.312	1 1/2	1 5/8	1 15/32	1 7/8	5/8	.197	5/16	.282	3/16	3/16	.040	.296	.003
3	.312	1 3/4	1 7/8	1 25/64	2 1/8	5/8	.197	5/16	.282	3/16	3/16	.040	.296	.003
3	.312	2	2 1/8	1 49/64	2 3/8	5/8	.197	5/16	.282	3/16	3/16	.040	.296	.003
4	.35	1 1/4	1 3/8	1 15/64	1 13/32	11/16	.228	11/32	.320	7/32	5/16	.050	.333	.003
4	.35	1 11/16	1 13/16	1 31/64	2 3/32	11/16	.228	11/32	.320	7/32	5/16	.050	.333	.003
5	.45	1 3/4	1 7/8	1 11/16	2 3/16	3/4	.260	3/8	.420	1/4	5/16	.060	.432	.003
5	.45	2	2 1/8	1 15/16	2 7/16	3/4	.260	3/8	.420	1/4	5/16	.060	.432	.003
5	.45	2 1/8	2 1/4	2 1/16	2 9/16	3/4	.260	3/8	.420	1/4	5/16	.060	.432	.003
6	.50	2 3/8	2 1/2	2 15/64	2 7/8	7/8	.291	7/16	.460	9/32	5/16	.060	.479	.005
6	.50	3 1/4	3 3/8	3 11/64	3 3/4	7/8	.291	7/16	.460	9/32	5/16	.060	.479	.005
7	.60	2 1/2	2 5/8	2 13/32	3 1/32	15/16	.322	15/32	.560	5/16	5/16	.070	.578	.005
7	.60	2 5/8	3	2 15/32	3 5/32	15/16	.322	15/32	.560	5/16	5/16	.070	.578	.005
7	.60	3	3 1/4	2 23/32	3 9/32	15/16	.322	15/32	.560	5/16	5/16	.070	.578	.005
7	.60	4	4 1/4	3 23/32	4 9/32	15/16	.322	15/32	.560	5/16	5/16	.070	.578	.005
8	.75	3 9/16	3 11/16	3 29/64	4 1/8	1	.353	1/2	.710	11/32	3/8	.080	.726	.005
8	.75	4	4 1/8	3 29/64	4 9/16	1	.353	1/2	.710	11/32	3/8	.080	.726	.005
9	.90	4	4 1/8	3 7/8	4 5/8	1 1/8	.385	9/16	.860	13/32	7/16	.100	.874	.005
9	.90	4 1/4	4 3/8	4 1/8	4 5/8	1 1/8	.385	9/16	.860	13/32	7/16	.100	.874	.005
10	1.0446	5	5 1/8	4 27/32	5 5/32	1 5/16	.447	21/32	1.010	7/16	7/16	.110	1.0137	.005
10	1.0446	5 11/16	5 13/16	5 5/32	6 3/32	1 5/16	.447	21/32	1.010	7/16	7/16	.110	1.0137	.005
10	1.0446	5 7/32	6 1/32	6 1/16	6 15/16	1 5/16	.447	21/32	1.010	7/16	7/16	.110	1.0137	.005
11	1.25	5 15/16	6 1/16	5 23/32	6 21/32	1 5/8	.447	21/32	1.210	7/16	1/2	.130	1.247	.005
11	1.25	6 3/4	6 7/8	6 19/32	7 5/32	1 5/16	.447	21/32	1.210	7/16	1/2	.130	1.247	.005
12	1.50	7 1/4	7 3/8	7 1/16	7 15/16	1 1/2	.510	3/4	1.460	1/2	1/2	.150	1.466	.005
13	1.75	7 3/4	7 7/8	7 9/16	8 9/16	1 1/2	.510	3/4	1.710	1/2	1/2	.170	1.716	.010
14	2.00	8 1/4	8 3/8	8 1/16	9 1/16	1 11/16	.572	27/32	1.960	9/16	5/8	.190	1.962	.010
15	2.25	8 3/4	8 7/8	8 17/32	9 21/32	1 11/16	.572	27/32	2.210	9/16	5/8	.210	2.212	.010
16	2.50	9 1/2	9 3/8	9	10 1/4	1 7/8	.635	15/16	2.450	5/8	1	.230	3.458	.010
17	2.75	9 3/4	9 7/8	.....	.....	.....	.....	.....	.....	.....	.....	.....	2.708	.....
18	3.00	10 1/4	10 3/8	.....	.....	.....	.....	.....	.....	.....	.....	.....	2.957	.....

Digitized by Google

JARNO TAPERS.

652. The Jarno taper system is .0600″ per foot (1 in 20) for all numbers, and the number of the taper determines all the other dimensions, as all parts are functions of the number. For example, No. 4 taper is $\frac{4}{10}$″ at small end, $\frac{4}{8}$″ at large end, and $\frac{4}{2}$″ (or 2″) in length.

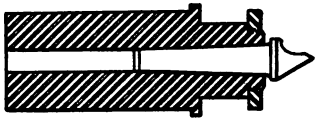


FIG. 431. — JARNO TAPER IN GRINDING-MACHINE SPINDLE.

As there is $\frac{1}{10}$″ in diameter between consecutive numbers at small end and $\frac{1}{8}$″ at large end, the number of a taper can be readily found by rough measurement with rule, or rule and calipers. Some prominent lathe grinding machine manufacturers have adopted the Jarno taper for holes in spindles, and others retain the Jarno taper per foot but modify the other specifications by changing the diameters or lengths to suit special conditions.

Digitized by Google

653. Table of Jarno tapers.

Taper per foot = .6 inch. Taper per inch = .050 inch.

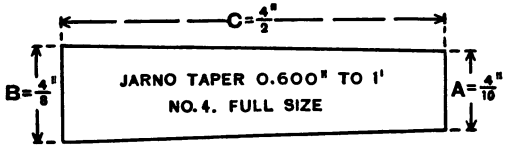


FIG. 432.

$$\text{Diameter of small end} = \frac{\text{No. of taper}}{10}.$$

$$\text{Diameter of large end} = \frac{\text{No. of taper}}{8}.$$

$$\text{Length of taper} = \frac{\text{No. of taper}}{2}.$$

NUMBER.	"A."	"B."	"C."
2	.250	.20	1
3	.375	.30	1½
4	.500	.40	2
5	.625	.50	2½
6	.750	.60	3
7	.875	.70	3½
8	1.000	.80	4
9	1.125	.90	4½
10	1.250	1.00	5
11	1.375	1.10	5½
12	1.500	1.20	6
13	1.625	1.30	6½
14	1.750	1.40	7
15	1.875	1.50	7½
16	2.000	1.60	8
17	2.125	1.70	8½
18	2.250	1.80	9
19	2.375	1.90	9½
20	2.500	2.00	10

Digitized by Google

654. Table of footstock setovers for Morse, and Brown & Sharpe tapers.

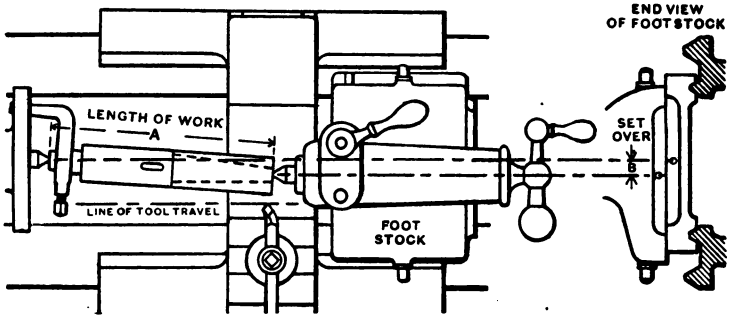


FIG. 433.

DIAGRAM AND FOOTSTOCK SETOVER FOR MORSE, AND BROWN & SHARPE TAPERS.

Digitized by Google

655. Table of footstock setovers for Morse, and Brown & Sharpe tapers. *Continued.*

Length of Stock in Inches.	The Amount in Inches of Setover of Footstock Given in Decimals and in Approximate Fractions.																	
	Setover for Morse Tapers.														Setover for Brown & Sharpe Tapers.			
	No. 0.		No. 1.		Nos. 2 and 3.		No. 4.		No. 5.		No. 6.		No. 7.		All Nos. Except No. 10.		No. 10.	
2	.0521	$\frac{3}{64}$	.0500	$\frac{3}{64}$	.0502	$\frac{3}{64}$	.0519	$\frac{3}{64}$	.....	.....	.....	.....	.....	.....	.0417	$\frac{3}{64}$	.....	...
3	.0781	$\frac{5}{64}$	.0750	$\frac{5}{64}$	.0752	$\frac{5}{64}$	.0779	$\frac{5}{64}$	.....	.....	.....	.....	.....	.....	.0625	$\frac{1}{16}$	.....	...
4	.1042	$\frac{7}{64}$	.1000	$\frac{7}{64}$	.1003	$\frac{7}{64}$	.1038	$\frac{7}{64}$	.1050	$\frac{7}{64}$	.....	.....	.....	.....	.0833	$\frac{5}{64}$	.....	...
5	.1302	$\frac{1}{8}$	.1250	$\frac{1}{8}$	.1254	$\frac{1}{8}$	.1298	$\frac{1}{8}$	.1313	$\frac{1}{8}$	.....	.....	.....	.....	.1042	$\frac{7}{64}$	.1075	$\frac{7}{64}$
6	.1563	$\frac{5}{32}$	.1500	$\frac{5}{32}$	.1505	$\frac{5}{32}$	.1558	$\frac{5}{32}$	.1575	$\frac{5}{32}$	.1565	$\frac{5}{32}$	.....	.....	.1250	$\frac{1}{8}$	.1290	$\frac{1}{8}$
7	.1823	$\frac{3}{16}$	.1750	$\frac{11}{64}$	.1756	$\frac{11}{64}$	.1817	$\frac{3}{16}$	.1838	$\frac{3}{16}$	.1826	$\frac{3}{16}$	.....	.....	.1458	$\frac{9}{64}$	.1505	$\frac{5}{32}$
8	.2083	$\frac{13}{64}$	.2000	$\frac{13}{64}$	.2007	$\frac{13}{64}$	.2077	$\frac{13}{64}$	.2100	$\frac{13}{64}$	.2087	$\frac{13}{64}$	.....	.....	.1667	$\frac{11}{64}$	.1720	$\frac{11}{64}$
9	.2344	$\frac{15}{64}$	.2250	$\frac{7}{32}$	.2258	$\frac{7}{32}$	.2336	$\frac{15}{64}$	.2363	$\frac{15}{64}$	.2348	$\frac{15}{64}$	.2344	$\frac{15}{64}$	.1875	$\frac{3}{16}$	.1935	$\frac{3}{16}$
10	.2604	$\frac{17}{64}$	.2500	$\frac{1}{4}$	.2508	$\frac{1}{4}$	.2596	$\frac{17}{64}$	.2625	$\frac{17}{64}$	.2608	$\frac{17}{64}$	.2604	$\frac{17}{64}$	.2083	$\frac{13}{64}$	.2150	$\frac{7}{32}$
11	.2865	$\frac{9}{32}$	.2750	$\frac{9}{32}$	.2759	$\frac{9}{32}$	.2855	$\frac{9}{32}$	.2888	$\frac{9}{32}$	.2869	$\frac{9}{32}$	.2865	$\frac{9}{32}$	.2292	$\frac{15}{64}$	.2365	$\frac{15}{64}$
12	.3125	$\frac{5}{16}$	.3000	$\frac{19}{64}$	.3010	$\frac{19}{64}$	.3115	$\frac{5}{16}$	.3150	$\frac{5}{16}$	.3130	$\frac{5}{16}$	.3125	$\frac{5}{16}$	.2500	$\frac{1}{4}$	.2581	$\frac{17}{64}$
13	.3385	$\frac{11}{32}$	.3250	$\frac{21}{64}$	.3261	$\frac{21}{64}$	.3375	$\frac{21}{64}$	.3413	$\frac{11}{32}$	.3391	$\frac{11}{32}$	.3385	$\frac{11}{32}$	.2708	$\frac{17}{64}$	.2796	$\frac{9}{32}$
14	.3646	$\frac{23}{64}$	.3500	$\frac{23}{64}$	.3512	$\frac{23}{64}$	.3634	$\frac{23}{64}$	.3675	$\frac{3}{8}$	.3652	$\frac{23}{64}$	.3646	$\frac{23}{64}$	.2917	$\frac{19}{64}$	.3011	$\frac{19}{64}$
15	.3906	$\frac{25}{64}$	.3750	$\frac{3}{8}$	.3763	$\frac{3}{8}$	.3894	$\frac{25}{64}$	.3938	$\frac{25}{64}$	.3913	$\frac{25}{64}$	.3906	$\frac{25}{64}$	.3125	$\frac{5}{16}$	.3226	$\frac{21}{64}$
16	.4167	$\frac{27}{64}$	.4000	$\frac{13}{32}$	.4013	$\frac{13}{32}$	.4153	$\frac{27}{64}$	.4200	$\frac{27}{64}$	.4173	$\frac{27}{64}$	.4167	$\frac{27}{64}$	.3333	$\frac{11}{32}$	.3441	$\frac{11}{32}$
17	.4427	$\frac{7}{16}$	.4250	$\frac{27}{64}$	.4264	$\frac{27}{64}$	.4413	$\frac{7}{16}$	.4463	$\frac{29}{64}$	.4434	$\frac{7}{16}$	.4427	$\frac{7}{16}$	.3542	$\frac{23}{64}$	.3655	$\frac{23}{64}$
18	.4688	$\frac{15}{32}$	.4500	$\frac{29}{64}$	.4515	$\frac{29}{64}$	.4673	$\frac{15}{32}$	.4725	$\frac{15}{32}$	.4695	$\frac{15}{32}$	.4688	$\frac{15}{32}$	.3750	$\frac{3}{8}$	.3871	$\frac{25}{64}$

Digitized by Google

Table of footstock setovers for Morse, and Brown & Sharpe tapers. *Concluded.*

Length of Stock in Inches.	The Amount in Inches of Setover of Footstock Given in Decimals and in Approximate Fractions.																	
	Setover for Morse Tapers.														Setover for Brown & Sharpe Tapers.			
	No. 0.		No. 1.		Nos. 2 and 3.		No. 4.		No. 5.		No. 6.		No. 7.		All Nos. Except No. 10.		No. 10.	
19	.4948	1/2	.4750	15/32	.4766	31/64	.4932	1/2	.4988	1/2	.4956	1/2	.4948	1/2	.3958	25/64	.4086	13/32
20	.5208	33/64	.5000	1/2	.5017	1/2	.5192	33/64	.5250	17/32	.5217	33/64	.5208	33/64	.4167	27/64	.4301	7/16
21	.5469	35/64	.5250	17/32	.5268	17/32	.5451	35/64	.5513	35/64	.5478	35/64	.5469	35/64	.4375	7/16	.4516	29/64
22	.5729	37/64	.5500	35/64	.5518	35/64	.5711	37/64	.5775	37/64	.5738	37/64	.5729	37/64	.4583	29/64	.4731	15/32
23	.5990	19/32	.5750	37/64	.5769	37/64	.5970	19/32	.6038	39/64	.5999	19/32	.5990	19/32	.4792	31/64	.4946	31/64
24	.6250	5/8	.6000	39/64	.6020	39/64	.6230	5/8	.6300	5/8	.6260	5/8	.6250	5/8	.5000	1/2	.5161	33/64
25	.6510	21/32	.6250	5/8	.6271	5/8	.6490	21/32	.6563	21/32	.6521	21/32	.6510	21/32	.5208	33/64	.5376	17/32
26	.6771	43/64	.6500	21/32	.6522	21/32	.6749	43/64	.6825	11/16	.6782	43/64	.6771	43/64	.5417	35/64	.5591	9/16
27	.7031	45/64	.6750	43/64	.6773	43/64	.7009	45/64	.7088	45/64	.7043	45/64	.7031	45/64	.5625	9/16	.5806	37/64
28	.7292	47/64	.7000	45/64	.7023	45/64	.7268	47/64	.7350	47/64	.7303	47/64	.7292	47/64	.5833	37/64	.6021	19/32
29	.7552	3/4	.7250	23/32	.7274	23/32	.7528	3/4	.7613	49/64	.7564	3/4	.7552	3/4	.6042	39/64	.6236	39/64
30	.7813	25/32	.7500	3/4	.7525	3/4	.7788	25/32	.7875	25/32	.7825	25/32	.7813	25/32	.6250	5/8	.6451	41/64
31	.8073	13/16	.7750	25/32	.7776	25/32	.8047	13/16	.8138	13/16	.8086	13/16	.8073	13/16	.6458	41/64	.6666	43/64
32	.8333	53/64	.8000	51/64	.8027	51/64	.8307	53/64	.8400	27/32	.8347	53/64	.8333	53/64	.6667	43/64	.6881	11/16
33	.8594	55/64	.8250	53/64	.8278	53/64	.8566	55/64	.8663	55/64	.8608	55/64	.8594	55/64	.6875	11/16	.7096	45/64
34	.8854	57/64	.8500	27/32	.8528	55/64	.8826	57/64	.8925	57/64	.8868	57/64	.8854	57/64	.7083	45/64	.7311	47/64
35	.9115	29/32	.8750	7/8	.8779	7/8	.9085	29/32	.9188	59/64	.9129	29/32	.9115	29/32	.7292	47/64	.7526	3/4
36	.9375	15/16	.9000	29/32	.9030	29/32	.9345	15/16	.9450	15/16	.9390	15/16	.9375	15/16	.7500	3/4	.7742	49/64

Attention. — Preferably use the decimals if in **black face type**; if not, use the common fractions as these are nearer correct.

Digitized by Google

656. Table of decimal equivalents of American Screw Company's machine and wood screws.

The difference between consecutive sizes is .01316".

No. of Screw Gage.	Size of Number in Decimals.	No. of Screw Gage.	Size of Number in Decimals.	No. of Screw Gage.	Size of Number in Decimals.
000	.03152	16	.26840	34	.50528
00	.04468	17	.28156	35	.51844
0	.05784	18	.29472	36	.53160
1	.07100	19	.30788	37	.54476
2	.08416	20	.32104	38	.55792
3	.09732	21	.33420	39	.57108
4	.11048	22	.34736	40	.58424
5	.12364	23	.36052	41	.59740
6	.13680	24	.37368	42	.61056
7	.14996	25	.38684	43	.62372
8	.16312	26	.40000	44	.63688
9	.17628	27	.41316	45	.65004
10	.18944	28	.42632	46	.66320
11	.20260	29	.43948	47	.67636
12	.21576	30	.45264	48	.68952
13	.22892	31	.46580	49	.70268
14	.24208	32	.47896	50	.71584
15	.25524	33	.49212	.....	.....

Digitized by Google

657. Table of standard mandrel dimensions. Fig. 434.

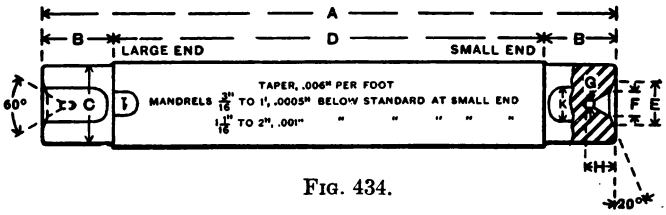


FIG. 434.

Nominal Diam.	Total Length.	Length of Ends.	Diam. of Ends.	Length of Taper.	Diam. of Recess.	Diam. of Countersinks.	Drill Size Fraction and Number.		Depth of Drilled Hole.	Width of Flat.
	A	B	C	D	E	F	G		H	K
3/16	3 1/4	5/16	5/32	2 5/8	.....	5/64	.....	56	5/32	1/8
1/4	3 3/4	7/16	7/32	2 7/8	11/64	1/8	1/16	52	7/32	5/32
5/16	4	7/16	9/32	3 1/8	7/32	1/8	1/16	52	7/32	3/16
3/8	4 1/4	1/2	11/32	3 1/4	1/4	5/32	1/16	52	1/4	7/32
7/16	4 1/2	1/2	13/32	3 1/2	1/4	5/32	1/16	52	1/4	1/4
1/2	5	9/16	15/32	3 7/8	5/16	3/16	5/64	47	9/32	1/4
9/16	5 1/4	9/16	17/32	4 1/8	5/16	3/16	5/64	47	9/32	9/32
5/8	5 1/2	5/8	19/32	4 1/8	13/32	7/32	5/64	47	5/16	9/32
11/16	5 3/4	5/8	21/32	4 1/2	3/8	7/32	5/64	47	5/16	5/16
3/4	6	3/4	23/32	4 1/2	7/16	1/4	3/32	42	3/8	5/16
13/16	6 1/4	3/4	25/32	4 3/4	15/32	1/4	3/32	42	3/8	5/16
7/8	6 1/2	13/16	27/32	4 7/8	17/32	9/32	3/32	42	13/32	3/8

Digitized by Google

Table of standard mandrel dimensions. *Concluded.*

Nominal Diam.	Total Length.	Length of Ends.	Diam. of Ends.	Length of Taper.	Diam. of Recess.	Diam. of Countersinks.	Drill Sizes Fraction and Number.		Depth of Drilled Hole.	Width of Flat.
	A	B	C	D	E	F	G		H	K
15/16	6 3/4	7/8	29/32	5	17/32	9/32	3/32	42	13/32	3/8
1	7	7/8	31/32	5 1/4	9/16	5/16	3/32	42	7/16	3/8
1 1/16	7 1/4	7/8	1	5 1/2	9/16	5/16	3/32	42	7/16	7/16
1 1/8	7 1/2	1	1 1/16	5 1/2	9/16	5/16	7/64	38	7/16	7/16
1 3/16	7 3/4	1	1 1/8	5 3/4	5/8	11/32	7/64	38	15/32	7/16
1 1/4	8	1	1 3/16	6	5/8	11/32	7/64	38	15/32	1/2
1 5/16	8 1/4	1	1 1/4	6 1/4	11/16	3/8	7/64	38	1/2	1/2
1 3/8	8 1/2	1 1/8	1 5/16	6 1/4	11/16	3/8	7/64	38	1/2	1/2
1 7/16	8 3/4	1 1/8	1 3/8	6 1/2	3/4	3/8	7/64	38	1/2	9/16
1 1/2	9	1 1/8	1 7/16	6 3/4	3/4	3/8	1/8	28	1/2	9/16
1 9/16	9 1/4	1 1/8	1 1/2	7	13/16	13/32	1/8	28	17/32	9/16
1 5/8	9 1/2	1 3/16	1 9/16	6 3/8	13/16	13/32	1/8	28	17/32	5/8
1 11/16	9 3/4	1 3/16	1 5/8	6 5/8	7/8	13/32	1/8	28	17/32	5/8
1 3/4	10	1 3/16	1 11/16	6 7/8	7/8	13/32	1/8	28	9/16	5/8
1 13/16	10 1/4	1 1/4	1 3/4	7 1/4	7/8	7/16	1/8	28	19/32	11/16
1 7/8	10 1/2	1 1/4	1 13/16	8	15/16	7/16	1/8	28	19/32	11/16
1 15/16	10 3/4	1 1/4	1 7/8	8 1/4	15/16	7/16	1/8	28	19/32	11/16
2	11	1 1/4	1 15/16	8 1/2	15/16	7/16	1/8	28	19/32	3/4

Digitized by Google

658. Number of revolutions required to obtain surface speeds of from 20 feet to 100 feet per minute.

DIAMETERS FROM ½ INCH TO 24 INCHES.

FEET PER MIN.	20	25	30	35	40	45	50	55	60	65	70
DIAM., INCHES.	REVOLUTIONS PER MINUTE.										
½	152	191	229	267	305	344	382	420	458	496	535
⅝	101	127	152	178	203	228	254	279	305	330	356
1	76	95	114	133	152	172	191	210	229	248	267
1⅛	68	85	102	119	136	153	170	187	204	221	238
1¼	61	76	91	106	122	137	153	168	183	199	213
1½	50	63	76	89	101	114	127	139	152	165	178
1¾	43	54	65	76	87	98	109	120	131	141	152
2	38	47	57	66	76	86	95	105	114	124	133
3	25	31	38	44	50	57	63	70	76	82	89
4	19	23	28	33	38	43	47	52	57	62	66
5	15	19	22	26	30	34	38	42	45	49	53
6	12	15	19	22	25	28	31	34	38	41	44
7	10	13	16	19	21	24	27	30	32	35	38
8	9	11	14	16	19	21	23	26	28	31	33
9	8	10	12	14	17	19	21	23	25	27	29
10	7	9	11	13	15	17	19	21	22	24	26
11	6	8	10	12	13	15	17	19	20	22	24
12	6	8	9	11	12	14	15	17	19	20	22
13	5	7	8	10	11	13	14	16	17	19	20
14	5	6	8	9	10	12	13	15	16	17	19
15	5	6	7	8	10	11	12	14	15	16	17
16	4	6	7	8	9	10	11	13	14	15	16
17	4	5	6	7	8	10	11	12	13	14	15
18	4	5	6	7	8	9	10	11	12	13	14
19	4	5	6	7	8	9	10	11	12	13	14
20	3	4	5	6	7	8	9	10	11	12	13
21	3	4	5	6	7	8	9	10	10	11	12
22	3	4	5	6	6	7	8	9	10	11	12
23	3	4	4	5	6	7	8	9	10	10	11
24	3	4	4	5	6	7	7	8	9	10	11

Digitized by Google

Number of revolutions required to obtain surface speeds of from 20 feet to 100 feet per minute. *Concluded.*

DIAMETERS FROM ½ INCH TO 24 INCHES.

FEET PER MIN.	75	80	85	90	95	100	105	110	115	120	125
DIAM., INCHES.	REVOLUTIONS PER MINUTE.										
½	573	611	649	687	726	764	802	840	878	916	955
¾	381	406	432	457	482	508	535	560	586	612	637
1	286	305	324	344	363	382	401	420	439	458	477
1⅛	255	272	289	306	323	340	357	374	391	408	425
1¼	228	245	260	274	290	306	321	336	351	367	382
1½	191	203	216	229	241	254	266	280	292	306	318
1¾	163	174	185	196	207	218	238	240	250	262	273
2	143	152	162	172	181	191	200	210	219	229	238
3	95	101	108	114	121	127	133	140	146	153	159
4	71	76	81	86	90	95	100	105	109	114	119
5	57	61	64	68	72	76	80	84	87	91	95
6	47	51	54	57	60	63	67	70	73	76	79
7	40	43	46	49	51	54	57	60	62	65	68
8	35	38	40	42	45	47	50	52	55	57	59
9	31	34	36	38	40	42	44	46	48	51	53
10	28	30	32	34	36	38	40	42	44	46	47
11	26	27	29	31	33	34	36	38	40	41	43
12	23	25	27	28	30	31	33	35	36	38	39
13	22	23	25	26	27	29	30	32	33	35	36
14	20	21	23	24	25	27	28	30	31	32	34
15	19	20	21	23	24	25	26	28	29	30	31
16	17	19	20	21	22	23	25	26	27	28	29
17	16	17	19	20	21	22	23	24	25	26	28
18	15	16	18	19	20	21	22	23	24	25	26
19	15	16	17	18	19	20	21	22	23	24	25
20	14	15	16	17	18	19	20	21	22	22	23
21	13	14	15	16	17	18	19	20	20	21	22
22	13	13	14	15	16	17	18	19	19	20	21
23	12	13	14	15	15	16	17	18	19	19	20
24	11	12	13	14	15	16	16	17	18	19	19

Attention. — To calculate any cutting speed, see § 96.

Digitized by Google

659. A comparative table of different wire gage sizes in general use in the United States.

IN DECIMAL PARTS OF AN INCH.

Number of Wire Gage.	American or Brown & Sharpe.	Birmingham, or Iron, Stubs' Wire.	American Steel & Wire Co.'s Gage.	Imperial Wire Gage.	Trenton Iron Co.	Stubs' Steel Wire.	U. S. Stand. for Plate.	Number of Wire Gage.
000000	.....	.....	.....	.464	.....	.....	.46875	000000
00000	.....	.....	.....	.432	.45	.....	.4375	00000
0000	.46	.454	.3938	.400	.4	.....	.40625	0000
000	.40964	.425	.3625	.372	.36	.....	.375	000
00	.3648	.38	.3310	.348	.33	.....	.34375	00
0	.32486	.34	.3065	.324	.305	.....	.3125	0
1	.2893	.3	.2830	.300	.285	.227	.28125	1
2	.25763	.284	.2625	.276	.265	.219	.265625	2
3	.22942	.259	.2437	.252	.245	.212	.25	3
4	.20431	.238	.2253	.232	.225	.207	.234375	4
5	.18194	.22	.2070	.212	.205	.204	.21875	5
6	.16202	.203	.1920	.192	.19	.201	.203125	6
7	.14428	.18	.1770	.176	.175	.199	.1875	7
8	.12849	.165	.1620	.160	.16	.197	.171875	8
9	.11443	.148	.1483	.144	.145	.194	.15625	9
10	.10189	.134	.1350	.128	.13	.191	.140625	10
11	.090742	.12	.1205	.116	.1175	.188	.125	11
12	.080808	.109	.1055	.104	.105	.185	.109375	12
13	.071961	.095	.0915	.092	.0925	.182	.09375	13
14	.064084	.083	.0800	.080	.08	.180	.078125	14
15	.057068	.072	.0720	.072	.07	.178	.0703125	15
16	.05082	.065	.0625	.064	.061	.175	.0625	16
17	.045257	.058	.0540	.056	.0525	.172	.05625	17
18	.040303	.049	.0475	.048	.045	.168	.05	18
19	.03589	.042	.0410	.040	.04	.164	.04375	19
20	.031961	.035	.0348	.036	.035	.161	.0375	20
21	.028462	.032	.03175	.032	.031	.157	.034375	21
22	.025347	.028	.0286	.028	.028	.155	.03125	22
23	.022571	.025	.0258	.024	.025	.153	.028125	23
24	.0201	.022	.0230	.022	.0225	.151	.025	24
25	.0179	.02	.0204	.020	.02	.148	.021875	25
26	.01594	.018	.0181	.018	.018	.146	.01875	26
27	.014195	.016	.0173	.0164	.017	.143	.0171875	27
28	.012641	.014	.0162	.0149	.016	.139	.015625	28
29	.011257	.013	.0150	.0136	.015	.134	.0140625	29
30	.010025	.012	.0140	.0124	.014	.127	.0125	30
31	.008928	.01	.0132	.0116	.013	.120	.0109375	31
32	.00795	.009	.0128	.0108	.012	.115	.01015625	32
33	.00708	.008	.0118	.0100	.011	.112	.009375	33
34	.006304	.007	.0104	.0092	.01	.110	.00859375	34
35	.005614	.005	.0095	.0084	.0095	.108	.0078125	35
36	.005	.004	.0090	.0076	.009	.106	.00703125	36
37	.004453	.....	.....	.0068	.0085	.103	.006640625	37
38	.003965	.....	.....	.0060	.008	.101	.00625	38
39	.003531	.....	.....	.0052	.0075	.099	.....	39
40	.003144	.....	.....	.0048	.007	.097	.....	40

Digitized by Google

660. Table of decimal equivalents of Stubs' steel wire gage.

Letter	Sizes of Letters in Decimals.	No. of Wire Gage.	Sizes of Numbers in Decimals.	No. of Wire Gage.	Sizes of Numbers in Decimals.	No. of Wire Gage.	Sizes of Numbers in Decimals.
Z	.413	1	.227	28	.139	55	.050
Y	.404	2	.219	29	.134	56	.045
X	.397	3	.212	30	.127	57	.042
W	.386	4	.207	31	.120	58	.041
V	.377	5	.204	32	.115	59	.040
U	.368	6	.201	33	.112	60	.039
T	.358	7	.199	34	.110	61	.038
S	.348	8	.197	35	.108	62	.037
R	.339	9	.194	36	.106	63	.036
Q	.332	10	.191	37	.103	64	.035
P	.323	11	.188	38	.101	65	.033
O	.316	12	.185	39	.099	66	.032
N	.302	13	.182	40	.097	67	.031
M	.295	14	.180	41	.095	68	.030
L	.290	15	.178	42	.092	69	.029
K	.281	16	.175	43	.088	70	.027
J	.277	17	.172	44	.085	71	.026
I	.272	18	.168	45	.081	72	.024
H	.266	19	.164	46	.079	73	.023
G	.261	20	.161	47	.077	74	.022
F	.257	21	.157	48	.075	75	.020
E	.250	22	.155	49	.072	76	.018
D	.246	23	.153	50	.069	77	.016
C	.242	24	.151	51	.066	78	.015
B	.238	25	.148	52	.063	79	.014
A	.234	26	.146	53	.058	80	.013
.....	.....	27	.143	54	.055	.....	.....

Digitized by Google

661. Table of different steel music wire gage in decimals of an inch.

AMERICAN STEEL & WIRE CO.

Number of Gage.	American S. & W. Co's. Steel and Iron Wire Gage	New American S. & W. Co's. Music Wire Gage.	Washburn Wire Co.	Trenton Iron Co.	Poehlmann Gage.	Felten & Guilleaume.	English Wire Gage.	Webster & Horsefall.
6.0	.4615	.004	.....	.....	.....	.....	.....	.....
5.0	.430	.005	.....	.....	.....	.....	.....	.....
4.0	.3938	.006	.007	.....	.006	.0068	.....	.006
3.0	.3625	.007	.0075	.....	.007	.0075	.....	.007
2.0	.331	.008	.0085	.008	.008	.0087	.....	.008
1.0	.3065	.009	.009	.009	.009	.0093	.....	.009
1	.283	.010	.010	.010	.010	.0098	.....	.010
2	.2625	.011	.011	.011	.011	.0106	.0105	.011
3	.2437	.012	.012	.012	.012	.0014	.0115	.012
4	.225	.013	.013	.013	.013	.0122	.0125	.013
5	.207	.014	.014	.014	.014	.0138	.0145	.014
6	.192	.016	.016	.016	.016	.0157	.015	.016
7	.177	.018	.018	.018	.018	.0177	.0175	.018
8	.162	.020	.020	.020	.020	.0197	.019	.020
9	.1483	.022	.022	.022	.022	.0216	.022	.022
10	.135	.024	.024	.024	.024	.0236	.0245	.024
11	.1205	.026	.026	.026	.026	.026	.027	.026
12	.1055	.029	.028	.029	.029	.0283	.0285	.029
13	.0915	.031	.030	.031	.031	.0303	.0305	.031
14	.080	.033	.032	.033	.033	.0323	.032	.033
15	.072	.035	.034	.035	.035	.0342	.035	.035
16	.0625	.037	.036	.037	.037	.0362	.036	.037
17	.054	.039	.038	.039	.039	.0382	.038	.039
18	.0475	.041	.040	.041	.041	.0400	.040	.041
19	.041	.043	.042	.043	.043	.042	.042	.043
20	.0348	.045	.044	.045	.045	.044	.043	.045
21	.03175	.047	.046	.047	.047	.046	.0445	.047
22	.0286	.049	.048	.049	.049	.048	.047	.052
23	.0258	.051	.051	.051	.051	.051	.049	.055
24	.023	.055	.055	.055	.055	.055	.053	.059
25	.0204	.059	.059	.059	.059	.059	.056	.061
26	.0181	.063	.063	.063	.063	.063	.0605	.065
27	.0173	.067	.067	.067	.067	.067	.064	.070
28	.0162	.071	.071	.071	.071	.071	.0685	.072
29	.015	.075	.074	.075	.075	.074	.0715	.077
30	.014	.080	.078	.080	.080	.078	.075	.083
31	.....	.085	.082	.085	.....	.082	.....	.....
32	.....	.090	.086	.090	.....	.086	.....	.....
33	.....	.095	.090	.095	.....	.....	.....	.....

Digitized by Google

662. Table of decimal equivalents of United States Standard sheet and plate steel and iron gage sizes.

Number of Gage.	Approximate Thickness in Fractions of an Inch.	Approximate Thickness in Decimal Parts of an Inch.	Weight per Square Foot in Ounces Avoirdupois.	Weight per Square Foot in Pounds Avoirdupois.
0000000	1-2	.5	320	20.00
000000	15–32	.46875	300	18.75
00000	7–16	.4375	280	17.50
0000	13–32	.40625	260	16.25
000	3–8	.375	240	15.00
00	11–32	.34375	220	13.75
0	5–16	.3125	200	12.50
1	9–32	.28125	180	11.25
2	17–64	.265625	170	10.625
3	1–4	.25	160	10.00
4	15–64	.234375	150	9.375
5	7–32	.21875	140	8.75
6	13–64	.203125	130	8.125
7	3–16	.1875	120	7.5
8	11–64	.171875	110	6.875
9	5–32	.15625	100	6.25
10	9–64	.140625	90	5.625
11	1–8	.125	80	5.00
12	7–64	.109375	70	4.375
13	3–32	.09375	60	3.75
14	5–64	.078125	50	3.125
15	9–128	.0703125	45	2.8125
16	1–16	.0625	40	2.5
17	9–160	.05625	36	2.25
18	1 20	.05	32	2.
19	7–160	.04375	28	1.75
20	3–80	.0375	24	1.50
21	11–320	.034375	22	1.375
22	1–32	.03125	20	1.25
23	9–320	.028125	18	1.25
24	1–40	.025	16	1.
25	7–320	.021875	14	.875
26	3–160	.01875	12	.75
27	11–640	.0171875	11	.6875
28	1–64	.015625	10	.625
29	9–640	.0140625	9	.5625
30	1–80	.0125	8	.5
31	7–640	.0109375	7	.4375
32	13–1280	.01015625	6 1–2	.40625
33	3–320	.009375	6	.375
34	11–1280	.00859375	5 1–2	.34375
35	5–640	.0078125	5	.3125
36	9–1280	.00703125	4 1–2	.28125
37	17–2560	.006640625	4 1–4	.265625
38	1–160	.00625	4	.25

Digitized by Google

663. Uses of different wire and plate gage sizes. — The American or Brown & Sharpe gage is used for copper and electric wire; Birmingham or Stubs' iron wire gage, for machine steel and iron wire, also for sheet steel, iron, copper, brass, and seamless tubing; American Steel & Wire Company's gage, for steel wire; Imperial wire gage, for foreign work; Trenton Iron Company's gage, for their own work; Stubs' steel wire gage, for tool steel and drill rods; music (piano) wire (a tempered steel wire), in machine construction for coil and other springs. The new gage sizes run in even thousandths so that a micrometer may be used easily.

United States Standard plate gage is used for steel plate.

Attention. — As wire and sheet-metal fixed gages are not very accurate, it is best to use dimensions, not numbers, and to measure with a micrometer.

Digitized by Google

INDEX

A

Page

Abrasives 106
Accurate drilling 242–247
hand tapping 274, 275
Accurate drilling, inspection circle for 244
Acme standard or Twenty-nine degree thread 325–335
Adjustable reamers 141, 142
taps 271, 313
Alinement drilling and tapping 337, 338
of dead center 18, 19
Alining crankshaft center fixtures 341, 342
Aluminium, to machine 187
Alundum grinding wheels 198–202
Allowances for driving fits 70
forcing fits 71
grinding 207
reamed, ground, and bored holes 75, 76
reaming 139
running fits 69
shrinking fits 75
sliding fits 70
A. L. A. M. standard screws (bolts) and nuts 346, 347
American Screw Company's machine screws 297, 298, 363
A. S. M. E. standard machine screws 295–297
Ampere 344
Angle iron or angle plate, boring with 133
drilling with 250, 251
Angles, cutting 27, 28
pron, lathe 5–7
Approximate drilling 243
tapping 272–274
Arbor block 148
or mandrel press 74, 148, 149
Arbors or mandrels 146–154
Attachment, automatic tapping ... 266, 267
taper 84, 85
Attachments for lathes, 8, 17, 84, 85, 192, 194
Automatic dies 280
drill chuck and collet ... 252, 253
drift 233
sizing grinder 218, 219
taps 271
Automobile bolts. *See* Automobile screws
screws and nuts. 286, 346, 347

B

Babbitt, machining 187
Back gears, lathe 2–5
Back lash 114
rests for grinding machines .. 217–219

Page

Backing plate with a tap, to finish thread 312–314
Bearings, lubricating or oiling 13
treatment of roughed-up .. 13
Belts, putting on and pushing off 13, 14
Binding post, schedule of operations for making 183–187
Birmingham or Stubs' iron wire gage sizes 368, 372
Blocks, V 251
Body drills 236
Body drills, A. S. M. E. standard and special machine screw 296, 297
American Screw Company's machine screw . 298
Boiler tube holes, cutter for drilling 254, 255
Bolt and nut making 121–125
Bolt heads and nuts, table and formulas for 303, 348, 349
Bolt and nut, schedule of operations for making finished 124, 125
Bolts and nuts, automobile. *See* Automobile screws.
finished, 124, 125, 283, 348, 349
rough 283, 348, 349
semi-finished, 124, 125, 283, 284, 346–349
table of United States standard 348
Bolts, classes of 282, 283
drilling for 248, 260–269, 300
expansion 285, 290
finished 124, 125, 283
hanger 290
headless 285
hexagonal-head 282, 283
improvised 283, 284
length of thread on 282
machine screw 293
measuring 282
planer-head 283, 284
square-head 248, 283
stud 266, 267, 283, 284
tap 288
uses of 248, 282
Bolts and screws, spot facing for heads of 300
Bore to setting of inside calipers, adjusting tool to 306
Boring in the engine lathe 306–310
in the speed lathe 178
tool holders and cutters 95, 310
with angle iron or angle plate. 133
Boring, squaring to shoulders in .. 309, 310
Boring tool, grinding 43
Boring tools 29, 95, 308–310
Boring tools, setting 309

Digitized by Google

Page

Box jigs for drilling, reaming, and tapping 263–265
Brass turning in lathe 182–187
wire 368, 372
Brass binding post, making 183–187
Brass finishing, hand tools for 174–176
Brass, front tool for 182, 183
grinding twist drill for 232
rough hand turning 174–176
round-nose tool for 182, 183
twist drills for 232, 233
Bridges in hollow castings 147
British Association standard threads 102
Broach reamers 144, 145
Bronze, grinding phosphor 221
machining 187
Brown & Sharpe tapers 356, 357
Burr when squaring, to remove 33, 34
Burnishing 176, 177
Burnt or hot centers 56
Bushing for counterbores 301
Bushing, making a 128

C

Calculating compound gearing for threading 115, 116
diameter of blank to mill or file square or hexagonal 303
distance to set over footstock for tapers 81, 82
simple gearing for threading 110, 111
speeds and feeds for drills 237–241
speeds and feeds for lathe work 47–50
Caliper gages 77
squares 66, 67
Caliper, ten-thousandth micrometer 67
Calipers, adjusting tool to bore to setting of inside 306
adjusting tool to turn to setting of outside 46
divider 23
inside 305, 306
large inside micrometer 307, 308
measuring with inside 305, 306
with inside micrometer 306–308
measuring with micrometer 61–64
measuring with spring 45–47
vernier 65, 66
reading inside micrometer, 306–308
vernier 66
micrometer 61, 62
vernier 65, 67
setting inside 305
outside 45
small inside micrometer 306, 307
spring 45–47
thread 104
vernier 65–67
Calipers to another, to transfer setting from one pair of 47
Cap, counterboring 301
Cap screws, collar 288
countersunk-head 288, 300
drilling for 248, 249, 264, 265, 300

Page

Cap screws, fillister-head 288, 289, 300
-head stud 288
flat-head 288, 300
hexagonal-head 287, 288, 300
oval-countersunk head 288, 300
fillister-head 288, 300
round-head 288, 300
square-head 248, 288
tapping for 248, 249
uses of 287, 288
Carborundum grinding wheels 198–202
Carriage, lathe 1–3
Castle nuts 286, 346, 347
Cast iron, diamond-point tool for finish turning 36, 37
diamond-point tool for rough turning 36
drilling 230
hard 231, 232
finish facing 153
squaring 33, 34, 96
large square-nose tool for finish turning 36, 37
lathe tools for 27–37
rough facing 152
squaring 31
round-nose tool for finish turning 36, 37
small square-nose tool for finish turning 37
Cast-iron roll, grinding 216
Castings, drilling 231, 248–251, 260–263, 301
Castings, shrinkage of 349
estimating weight of cored 49
Castings from wooden patterns, weight of 349
Cat head 192, 193
Catching the thread in screw cutting 118
Celluloid grinding wheels 198, 200
Center gage 15
holes in lathe work 19–26
mandrels 149
punch 23, 24
reamers 22, 24
Center, alinement of dead 18, 19
dead 1, 2, 15, 16
live 2, 3, 16–18
locating live 4, 16
table 256
V 256
Center drill, to remove broken 24
fixtures, eccentric 340, 341
Center-hole dimensions, table of 21
Center for pipe turning, revolving dead 147
Center for squaring, grooved dead 33, 34
Centering machine 25, 26
tool for chucking 135
Centering, squaring, and straight turning, schedule of operations for 57–59
Centering work, hand method of 23, 24
machine method of 25, 26
Centers, burnt or hot 56
grinding machine, 195–197, 204–206
grinding engine lathe 17
lathe 1, 2, 3, 16–18
mounting work on lathe, 20, 24, 54–56
testing angle of 15
truing engine lathe 16, 17

Digitized by Google

Page
Centers, truing speed lathe.....15–18
grinding machine..204, 205
Centers in alinement, setting grinding machine..... 206
Chamfering nut and bolt..... 123
Change gear mechanism for screw cutting and feeds, rapid..... 8, 9
Change gears.....2, 3, 110–112, 115–117
Chart of forged lathe tools.....28, 29
grinding wheels..... 199
Charts of lathe holders and cutters..94, 95
Chasers, hand.....173, 174
Chattering of tools..... 91
Check nuts.....285, 286
Chuck jaws..... 126
Chucking.....134–138
an eccentric..... 129
a pulley..... 138
in speed lathe.....170, 171
with drill holder and steady rest..... 136
with twist drills...135, 170, 171
Chucking, centering tool for..... 135
Chucking drills, flat.....136–138
reamers, flat..... 137
fluted.....143, 144
Chucking with a flat drill and chucking reamer in an engine lathe, schedule of operations for.....137, 138
Chucks.....126–131, 232, 294, 295
Chucks, care of.....130, 131
classes of..... 126
combination..... 129
draw-in.....130, 294, 295
drill.....127, 230, 232, 255–258
face-plate..... 130
independent..... 127
mounting and removing... 131
set screw.....232, 255
special..... 127
spring.....130, 294, 295
tapping backing plates of 312–314
truing and holding work in combination..... 129
truing and holding work in independent..... 127
truing and holding work in universal..... 128
universal..... 128
Chucks to machine spindles, attaching.....126, 311–314
Clamp dog..... 121
Clamp nut, making a..... 123
Clamps, polishing..... 162
Clearance of counterbores.....299, 300
dies..... 280
drills..... 225
hand tools..... 164
lathe tools.....27, 28
reamers..... 139
taps..... 271
Coil springs.....179, 181
Cold punched nuts..... 285
Collar cap screws..... 288
Collet, to grind taper..... 220
Combination chucks..... 129
grinding wheels..... 200
Combination chucks, to true and hold work in..... 129
Compound gearing for threads, calculating.....115, 116
Compression springs..... 179
Copper wire.....368, 372

Page
Copper, machining 187
polishing.....162, 163
Cored castings, estim weight of. 349
Corundum grinding wh 198–202
Cotter, spring.....286, 346, 347
Coulomb..... 344
Counterbore with inserted cutter.... 302
Counterbore, forged.....299, 300
Counterbores, clearance of.....299, 300
Counterboring a cap..... 301
for screw heads....299–303
Counterbored holes..... 300
Counterbores.....299, 301, 302
Counterbores, separate bodies and guide bushings for..... 301
Countershaft, engine-lathe..... 2–4
speed-lathe.....11, 12
Countersink for screw heads.....302, 303
Countersinks.....22–25, 302, 303
Countersunk holes, 19–21, 24, 25, 300, 302, 303
Counterweights on face plates.....133, 342
Counting threads.....104, 105, 113
Crank, laying out holes in engine..132, 133
Crankshaft turning.....340–343
Crankshaft center fixtures, schedule of operations for laying out two-throw 90°.....340, 341
Crankshaft center fixtures, alining 341, 342
Cross feed of lathe..... 5–7
Crowning or tapering pulleys.156, 157, 159
Curve turning.....190, 191
Cut meter..... 48
Cut, finishing..... 28
roughing..... 28
Cutters for tool holders.....93–96, 98
Cutting angles.....27, 28
double threads.....335, 336
square threads..... 336
feeds for drills..... 237
metal plate by drilling..... 253
off stock.....97, 98
off wire in a universal hand lathe.....177, 178
screws.....102–120, 124, 125, 172–174, 279–281, 294, 295
Cutting feeds, lathe..... 50
Cutting speeds for high-speed steel lathe tools..... 49
Cutting speeds for hand tools.....164
lathe tools.....47–50
measuring.....48, 49
Cutting-off tool holders and cutters.....93–95, 98
Cutting-off tool, breaking a..... 98
grinding.....40, 41
improved hand...177, 178
Cutting-off tools, lathe...29, 94, 95, 97, 98
Cutting tools, lathe.....27–37, 87–98
lubricants for.....50–52
theory of 27, 28
rake, clearance and cutting angles of lathe.27, 28
Cylinder heads, drilling and tapping engine.....262, 266, 267
Cylindrical grinding.....195–221

D

Dead center, alinement of.....18, 19
lathe.....1, 2, 15, 16
Decimal equivalents, table of common fractions and..... 64

Digitized by Google

Page

Depth of cuts for grinding..... 203
Diameter to turn blank to mill or file square or hexagonal..... 303
Diamond tool for truing emery wheels 204
Diamond-point tools for cast iron... 34–37
for steel or wrought iron 88, 90, 91
Diamond-point tools, grinding..... 40, 43
half..... 29, 93
left..... 93
Dies, automatic..... 280
clearance of..... 280
machine-screw.....294, 295
sharpening..... 280
threading .279–281, 294, 295, 326–336
Disks, reference..... 78
Divider calipers..... 23
Dividers, setting..... 242
using.....242, 243
Dog, clamp..... 121
square..... 121
Dogs for grinding machine.....195, 205
Dogs for grinding work of one diameter, end-driving.....205, 206
Dogs or drivers, lathe..... 20
Double threads.....335, 336
tool holders..... 96
Dowel pins..... 289
Draw-in chucks.....130, 294, 295
Drawing the drill.....243–247
Drawings, schedule-of-operations.. 54
Drill chucks.....127, 230, 232, 255–257
drifts or center keys....230, 231, 233
gages.....234–236
Drill, drawing the.....243–247
extension for..... 255
Drill chuck and collet, automatic..252, 253
drifts, automatic..... 233
gages, numbered.....235, 236
Drill grinder, twist.....227, 228
Drill sizes, consecutive and comparative tables of different 350–353
Drilling.....222–269
and counterboring duplicate parts.....268, 269
and tapping engine cylinder heads.....262. 266, 267
Drilling and tapping cross-feed screw nut, alinement.....337, 338
Drilling brass or composition.....232, 233
castings,.231, 248–251, 260–269, 300
cast iron..... 230
deep holes.....258, 259
diametrically through shaft, 251–253, 256
flanges.....260–264
flat or square work in speed lathe.....255–257
for bolts.....248, 260–269, 300
screws and studs, 248, 249, 260–269, 300
set screws.....158–160, 247
forgings.....254–259
glass..... 52
hard cast iron.....231, 232
hole in end of shaft parallel to axis in speed lathe..... 257
jigs.....260–269
machine frames.....243–251, 260
on a slanting surface..... 254
part of a hole..... 254
steel or wrought iron.....230, 231

Page

Drilling with flange jig..... 262
wood..... 232
work held against angle plate.....250, 251
work held by hand, or in vise, on drilling table..... 250
Drilling, accurate.....242–247
approximate..... 243
inspection circle for accurate..244
laying out work for, 242–244, 246–250
oil drills for deep.....257–259
rough..... 243
Drilling, use of lead holes for.....253, 254
Drilling boiler tube holes, cutter for.....254, 255
Drilling cored holes, three and four-groove twist drills for.....250, 251
Drilling, reaming and tapping, schedule of operations for.....246, 247
Drilling machine described, vertical.....222–224
Drilling machines, classes of..... 222
feeds of 222–224, 240, 241
multiple-spindle 262–264, 268, 269
radial.....265–268
speeds of, 222–224, 239–241
Drilling table, leveling work on..... 255
Drills for brass.....232, 233
cast iron 230–232 234, 237, 250, 251
hard steel.....231, 232
steel and wrought iron, 230–232, 234–237, 257–259
sheet metal.....232, 233
Drills, body..... 236
classes of.....234–237
clearances for..... 225
effect of bad grinding of...225, 226
flat chucking.....136, 138
hand method of grinding twist, 225, 226, 228, 229
high-speed steel.....225, 231, 232
machine method of grinding twist.....227, 228
oil.....257–259
reamer..... 237
round-shank, flat..... 232
speeds and feeds for.....237–241
straight and taper-shank twist 230
straight-groove..... 233
tables of speeds and feeds for carbon-steel.....237, 238
table of speeds and feeds for high-speed steel..... 239
tap..... 237
taper-shank twist.....230–232
thinning the point of twist.... 226
twist.....225–259
Drills twisted from flat bar, twist..231, 232
Drills for brass, grinding twist..... 232
Drills without stopping machine, changing.....252, 253
Driving fits..... 70
lathe work.....20, 21
Duplicate pieces, schedule of operations for making..... 54
Duplicating sizes, lathe stops for.... 8

Digitized by Google

E

Page
Eccentric center fixtures....340, 341
turning....338–343
Eccentric, turning engine....339, 340
Eccentric shaft, turning....338, 339
Elastic grinding wheels.... 200
Electrical units.... 344
Electrically-driven engine lathe.... 10
machine tools.... 9, 10
Emery cloth for polishing, order of applying....160, 161
Emery wheels....198–202
Emery wheels, diamond tool for truing.... 204
End-measuring rods.... 78
Engine lathe.... 1–7
Engine-lathe countershaft.... 2–4
gear headstock....313, 314
headstock.... 2–5
Engine lathe with rapid system of change gears.... 8, 9
work....54–60
Engine lathe, hand reaming in....142, 143
power reaming in...143–145
tapping in....312, 313
English standard (Whitworth) threads. 102, 119, 120
Expanding mandrels....147, 153
Expansion bolts....285, 290
of work while grinding. 207, 208
turning.... 56
Extension for drill.... 255
springs.... 179

F

Face-plate chucks.... 130
Face plates, uses of....20, 21, 131–134
Face-plate work, lathe axis indicator for....133, 134
Facing bar with inserted cutter.... 302
disk held in a chuck.... 127
small gear blank with double holder and cutter.... 96
small gear blank with two forged tools.... 96
Facing for bolts and screw heads, spot.... 300
Feed for cylindrical grinding.... 203
vertical drilling machines, hand and power, 222–224, 240, 241
Feed, cutting.... 50
Fiber, machining hard rubber or.... 187
Files, mill....59, 60
Filing lathe work....59, 60
with round and half-round files.... 60
Filing, speed for lathe....59, 60
Fillets of cast iron, scraping....154, 155
Fillister-head cap screws....288, 289, 300
Fillister-head stud cap screws.... 288
Finish facing cast iron.... 153
grinding.... 207
of work by grinding.... 207
reaming.... 139
square....33, 34
squaring cast iron....33, 34, 96
steel or wrought iron....87, 88, 96
with side tool....33, 34
graver.... 166
threading with a tap....312, 313
Finish turning.... 28
cast iron....36, 37
cast iron, large square-nose tool for.... 37
cast iron, round-nose tool for....36, 37
cast iron, small square-nose tool for.... 37
steel or wrought iron..90–92
with a graver....165–167
Finished bolts and nuts, 124, 125, 283, 348, 349
Finishing cut.... 28
Fits in machine construction.... 68
Fits, classes of.... 68
driving.... 70
forcing....71–74
grinding....76, 209–215
holes for....75, 76
press for forcing....72–74
pressure and allowance for running....68, 69
producing standard.... 76
schedule of operations for grinding forcing....212, 213
schedule of operations for grinding running....210, 211
schedule of operations for turning and filing drive....93, 99, 100
schedule of operations for turning and filing running....98, 99
shrinking....73, 75
sliding.... 70
table of driving.... 70
forcing.... 71
tables of running.... 69
shrinking.... 75
sliding.... 70
taper forcing.... 71
running.... 68
turning and filing.... 76
Fits with micrometer, producing standard.... 76
Fitting, straight turning and....85, 86
Fixed nuts.... 337
Fixtures, eccentric center....340, 341
Flange drilling....260–264
jigs....261–264
Flange, clamping lathe carriage to face large.... 152
finishing face and scraping cast-iron (lathe work)....153–156
rough facing cast-iron.... 152
schedule of operations for making cast-iron....154–156
scraping fillets of cast-iron . 154, 155
Flat chucking drills....136–138
fillister-head cap screws....288, 293
machine screws. 293–300
Flat drills, round shank.... 232
Flat-head cap screws....288, 300
machine screws 293, 298, 300
Fluted chucking reamers....143, 144
Follower rests....193, 194
Footstock set-overs for turning tapers, table of....360–362
Forcing fits....71–74
press.... 72–74
Forgings, drilling.... 254–259
Forming tools....29, 95, 190, 191
Formulas for bolt heads and nuts 303, 348, 349

Digitized by Google

Page
Formulas for International and French standard threads 345
metric threads..... 345
Sharp V-threads 105
Square threads 315, 316
Twenty-nine degree threads..... 326
United States standard threads.....106, 107
Whitworth (English) standard threads..119, 120
Four-groove twist drills for drilling cored holes.....250, 251
Fractional threads.....114–117
Frames, drilling machine.....243–251, 260

G

Gage for grinding and setting Twenty-nine degree thread tools.....328–330
Gage sizes for United States standard sheet and plate steel and iron..... 371
Gage, American Screw Company's machine-screw 298
center..... 15
Sharp V-thread 15, 108
Square threading tool angle .. 317
twist drill grinding..... 229
United States standard thread 109
Gage sizes, decimal equivalents of different steel music wire..... 370
Gage sizes in general use, different wire..... 368
Gages, caliper..... 77
end-measuring rod..... 78
hole.....77–79
limit.....78, 79
numbered drill.....235, 236
reference disks..... 78
special..... 79
standard cylindrical-plug and ring..... 77
standard and limit.....77–79
twist-drill.....234–236
wire.....234–236
Gear headstock, engine-lathe.....313, 314
Gears, change.....2, 3, 110–112, 115–117
Glass, drilling 52
Graver.....164–167
Graver, finish squaring with..... 166
finish turning with.....165–167
grinding..... 42
rough squaring with..... 166
rough turning with.....165, 166
Grinder, automatic sizing.....218, 219
universal tool.....43, 44
wet-tool.....42–44
Grinding boring tool..... 43
cast-iron roll.....216, 217
cutting-off tool.....40, 41
diamond-point tool.....40, 43
fits.....76, 209–215
graver..... 42
high-speed steel cutter..... 42
lathe centers..... 17
machine centers, 195–197, 204–206
phosphor-bronze taper bushing..... 221
round-nose hand tools..... 43
round-nose tools.....37, 38
side tool.....38, 39, 43, 44
slender shaft.....217, 218
square threading tools.....40, 41

Page
Grinding straight and taper on one piece.....219, 220
taper collet..... 220
tapers.....213–215, 219–221
to shoulder..... 215
Twenty-nine degree threading tools.....328–330
twist drills for brass..... 232
United States standard or Sharp V threading tools..41, 42
with back rests.....217–219
Grinding, allowances for..... 207
cylindrical.....195–221
depth of cuts for..... 203
direction of rotation of work and wheel for..... 203
expansion of work while..207, 208
feed for cylindrical..... 203
lubricants for..... 207
machine.....195–221
measuring tools for..... 208
methods of driving work for.....205, 206
rough and finish..... 207
setting swivel table for straight..... 206
setting swivel table for taper..... 206
speed of work for..... 203
the finish of work by..... 207
wet and dry.....206–208
width of face of wheel for.. 203
Grinding drills, hand method of..228, 229
machine method of 227, 228
Grinding forcing fits, schedule of operations for.....212, 213
Grinding on two dead centers, principle of..... 195
Grinding machine, universal.....196, 197
universal and tool, 211, 212
Grinding machines, care of..... 208
classes of..... 196
operating universal 209
plain.....215–220
truing centers of, 204, 205
Grinding of drills, effect of good and bad.....225, 226
Grinding running fits, schedule of operations for.....210, 211
Grinding standard mandrel, schedule of operations for.....213–215
Grinding tapers, method of..... 206
Grinding wheels.....198–202
Grinding wheels, alundum.....198–202
carborundum....198–202
celluloid.....198, 200
chart of..... 199
combination..... 200
corundum.....198–202
elastic.....200, 201
emery 198–202
grades of.....198, 201
mounting of..... 202
selection of.....201, 202
shapes of.....199, 201
silicate.....199, 200
table of speeds for.. 202
tanite.....198, 200
truing 203, 204
vitrified.....198, 199
vulcanite.....198, 200

Digitized by Google

Page
Grinding work of one diameter, end-driving dog for....205, 206
Ground work, seasoning of.... 208
Grooved dead center for squaring...33, 34
Guards, water....207, 209, 210

H

Half diamond-point tools.... 93
Hammer, lead.... 148
machinists'....23, 24
rawhide.... 148
Hammers for finished work, soft.... 148
Hand chasers....173, 174
cutting-off tools....177, 178
nurling.... 188
reamers....139–143
reaming in engine lathe....142, 143
reaming stand...141, 142
vertical drilling machine.... 141
reaming in vise.... 140
tapping....269–279, 293, 294
threading....279–281, 294, 295
turning....164–179
machine handle....167–170
tools for brass finishing....174–176
Hand taps, set of....269, 270
turning, templets for....167, 168
tools, clearance of.... 164
cutting speeds for.... 164
graver....164–166
round-nose....166, 167
Hand tools from old files, making..178, 179
Handles, machine....167–170
Hanger bolts.... 290
Headless bolts.... 285
screws. *See* Set Screws.
Height of lathe cutting tools..27–37, 87–97
Hexagonal head bolt....282, 283
cap screws..287, 288, 300
Hexagonal, calculating diameter to turn blank for milling or filing.... 303
High-speed steel drills....225, 231, 232
lathe tools 28, 90, 91, 93, 95
High-speed steel drills, speeds for... 239
High-speed steel lathe tools, cutting speeds for....49
Holders and cutters, lathe, 93–96, 310–312, 318
Hole gages....77–79
Hole diameters, table of limits of....75, 76
Holes, allowances for reamed, ground and bored....75, 76
counterbored.... 300
countersunk, 19–21, 24, 25, 300, 302, 303
depth of tapped....272, 273
reamed..139–145, 236, 237, 242–247
table of allowances and limits for standard.... 76
tapped, 171, 172, 236, 237, 246–248, 273–279, 296–299
Hollow oil drill for deep drilling.... 259
Hollow shafting, method of drilling.. 259
Hot centers, burnt or 56
Hot pressed nuts.... 285

I

Improvised bolts....283, 284
Independent chucks.... 127
Independent chucks, to true and hold work in.... 127

Page
Indexing in the engine lathe.... 304
Indicator, axis....133, 134
surface speed attachment for speed....48, 49
Inside calipers....305, 306
Inside calipers, measuring with...305, 306
setting.... 305
Inside micrometer calipers, large..307, 308
small..306, 307
Inside micrometer calipers, to measure with....306–308
Inside micrometer calipers, reading, 306–308
Square threading tools 29, 95, 319, 320
Square threading tools, setting. 319
squaring tool, squaring with, 309, 310
threading or screw cutting tools, setting United States standard or Sharp V.... 311
Inside threading or screw cutting tools, United States standard or Sharp V.... 29, 95, 310, 311
Inside turning or boring in the speed lathe.... 178
Inside Twenty-nine degree threading tools....95, 329, 330
Inside United States standard or Sharp V threading in the engine lathe....310–314
Interchangeable machine parts.... 260
International and French standard threads.... 345
Interrupted thread tap.... 313
Iron wire....368, 372
Iron, lathe cutting tools for cast....27–37
lathe tools for steel or wrought..87–98

J

Jarno tapers....358, 359
Jig vise....267, 268
Jig, solid....260, 269
Jigs for tapping....265, 275
Jigs, box....263–265
classes of drilling.... 260
drilling....260–269
example of improvised....260, 261
multiple.... 260
plate (flange)....261, 262
rotary.... 260
Joule.... 344
Journals. *See* Bearings.

K

Kerosene for machining aluminium .. 51
Keys, drill drifts or center...230, 231, 233
Knurling. *See* Nurling.

L

Lag screws....289, 290
Lands of reamers.... 139
taps.... 270
Lard oil....50, 51
Lathe apron.... 5–7
back gears.... 2–5
box for tools.... 53, 55
carriage.... 1–3
centers....1, 2, 3, 16–18
cross feed.... 5–7
dogs or drivers.... 20
holders and cutters, 93–96, 310–312, 318
long. feed 5–7

Digitized by Google

Page

Lathe mandrels....146–153
revolutions....48, 49
stops for duplicating sizes.... 8
tools for cast iron....27–37
steel or wrought iron 29, 87–98
with rapid change-gear mechanism.... 8, 9
Lathe, electrically driven engine.... 10
engine.... 1–7
speed....11, 12
Lathe centers, grinding.... 17
mounting work on, 20, 21, 54–56
truing engine....16, 17
truing speed....15–18
Lathe tools, chart of forged....28, 29
clearance of....27, 28
cutting angles of....27, 28
cutting speeds of....47–50
height of....30, 35, 36
high-speed steel, 28, 90, 91, 93–95
rake of....27, 28
Lathe work, axis indicator for....133, 134
center holes in....19–26
filing....59, 60
Lathes, attachments for, 8, 17, 84, 85, 192, 194
classes of.... 7
requirements for successful use of engine.... 15
swing of.... 7
Laying out two-throw 90° crankshaft center fixtures....340, 341
Laying out holes in engine crank..132, 133
work for drilling, 242–244, 246–250
Lead hammer.... 148
Lead holes....253, 254
of screw threads....104, 117
screws....2, 3, 6, 7, 103
Lead, machining.... 187
Left diamond-point tools.... 93
side tool.... 92
threads....114, 314
Leveling work on drilling table.... 255
Limit gages....78, 79
Line tapping. *See* Alinement drilling and tapping.
Lining lathe centers....18, 19
Live center....2, 3, 16–18
Locating live center.... 4, 16
Lock or check nuts....285, 286
Long. feed, lathe.... 5–7
Lubricants for cutting tools....50–52
grinding.... 207
Lubricating or oiling bearings.... 13

M

Machine grinding....195–221
nurling.... 189
oil.... 13
Machine, centering....25, 26
Machine handle, schedule of operations for making a....168–170
Machine parts, interchangeable.... 260
Machine screw bolt.... 293
dies....294, 295
taps....293, 294
Machine screws....292, 293, 295–299, 363

Page

Machine screws, A. S. M. E. standard and special 295, 297
American Screw Company's....297, 298, 363
flat-head.... 293
fillister-head.... 293
oval countersunk head.... 293
round-head.... 293
Machinists' hammer....23, 24
Machines, care of.... 53
Mandrel block.... 148
or arbor press....74, 148, 149
Mandrel, schedule of operations for making standard....149–152
schedule of operations for grinding standard....213–215
soft hammers for driving... 148
Mandrel dimensions, standard...364, 365
Mandrel into work with a press, forcing....148, 149
Mandrels or arbors....146–154
Mandrels or arbors, built-up and special, 147, 148
bridges in hollow castings to take place of.... 147
center.... 149
expanding....147, 153
lathe....146–153
nut.... 122
standard solid, 146, 349–352, 364, 365
Master taps.... 271
Material (stock), inspection and measurement of.... 52
Measuring with spring calipers....45–47
thread calipers.... 104
work with inside calipers, 305, 306
micrometer calipers....61–63
outside calipers 45–47
vernier calipers, 65, 66
Metal to wood, fastening.... 291
Meter, cut.... 48
Metric threads....102, 117, 345
Micrometer caliper, ten-thousandth.. 67
Micrometer calipers, large inside..307, 308
measuring with, 61–64
measuring with inside....306–308
reading....61, 62
small inside..306, 307
Micrometer principle.... 61
Mill files....59, 60
Milk for machining copper.... 51
Morse tapers....354, 355
Mounting work on lathe centers, 20, 21, 54–56
Multiple jigs.... 260
Multiple-spindle drilling machines 262–264, 268, 269
Multiple threads....335, 336
Multiple-threaded taps.... 336
Music wire gage sizes in decimals of an inch, table of different steel.... 370

N

Numbered drill gages....235, 236
Nurling tools....188, 189
Nurling, hand.... 188

Digitized by Google

Page
Nurling machine 189
Nurled head screws..183–186, 188, 189, 292
Nut and bolt making 121–125
mandrels 122
taps 271
Nut, fitting Square thread screw to.. 325
fitting Twenty-nine degree thread screw to 332, 333
fitting United States standard or Sharp V thread screw to. 114, 125
making Square thread 323–325
Twenty-nine degree thread...333–335
Nut in alinement, drilling and tapping a cross feed 337, 338
Nuts, automobile standard screws and 286, 346, 347
bracket 337
bushing 337
castle 286, 346, 347
clamp 123
cold punched 285
description of 285
fixed 337
hot pressed 285
lock or check 285, 286
planer 284
threading, 102–120, 173, 174, 183, 185, 186, 269–279, 292–294, 310, 338
thumb 292
wing 292

O

Ohm 344
Oil, kerosene 51
lard 50, 51
lubricating 13
sperm 50
Oil drills for deep drilling 257–259
Oiling bearings 13
ways of a machine 13
Oil stones, Arkansas 44
India 44
Oilstoning tools 44
Order of operations. *See* Schedule of operations.
Operations. *See* Schedule of operations.
Oval countersunk-head cap screws..288, 300
countersunk-head machine screws 293, 300
fillister-head cap screws.....288, 300
head machine screws.293, 300

P

Pin, split 286, 346, 347
Pins, dowel 289
taper 144, 145
Pipe tap, threading 119
Pipe turning, revolving dead center for 147
Pitch of threads 104
Plain grinding machines 215–220
Planer nuts 284
Planer head bolts 283, 284
Planisher, turning brass with 175, 176
Plate jigs 261, 262
Plate and wire gages, uses of different 372
Plate steel and iron gage sizes, United States standard sheet and 371
Polishing brass 162, 163
cast-iron flange 160, 161

Page
Polishing copper 162, 163
clamps 162
steel shaft 162
Polishing, abrasives, speeds, and machines to use for 160
order of applying emery cloth for 160, 161
Power reaming in engine lathe....143–145
tapping 266, 267, 277
threading 280
Press fits. *See* Forcing fits.
Press, forcing 72, 74
mandrel or arbor 148, 149
Pulley taps 271
Pulley, chucking 137, 138
schedule of operations for making 158–160
Pulleys, locating set screws for....158–160
tapering or crowning 156–159
Punch, center 23, 24

R

Radial drilling machines 265–268
Rake of lathe tools 27, 28
Rawhide hammer 148
Rawhide, machining 187
Reading inside micrometer calipers, 306–308
vernier calipers 66
Reading micrometer calipers 61, 62
vernier calipers 65, 67
Reamed holes..139–145, 236, 237, 242–247
Reamer drills 237
Reamers, adjustable 141, 142
broach 144, 145
center 23, 24
clearance of 139
flat chucking 137
fluted chucking 143, 144
fluted taper pin 144, 145
hand 139–143
irregularly spaced teeth on 139
lands of 139
rose chucking 144
Reaming taper holes in a speed lathe by power 144, 145
Reaming, allowances for 139
finish 139
rough 139
Reaming in engine lathe, hand.....142, 143
power....143–145
Reaming in reaming stand, hand..141, 142
Reaming in vertical drilling machine, hand 141
Reaming in vise, hand 140
Reference disks 78
Rests, follower 193, 194
steady 191–193
Revolutions of lathe 48, 49
Ripper, rough turning brass with a, 174, 175
Rolled threads 103
Rotary jigs 260
Rough bolts and nuts 283, 348, 349
drilling 243
grinding 207
reaming 139
squaring 33
cast iron 31
steel or wrought iron 87, 88
with graver 166
with side tool, 33, 34, 87, 88
turning 28
cast iron 36

Digitized by Google

Page

Rough turning steel or wrought iron..88–90
with graver.....165, 166
Roughed-up bearings, treatment of.. 13
Roughing cut..... 28
Roughing tool, large.....89, 90
small.....29, 89
Round-head cap screws.....288, 300
machine screws.....293, 300
Round-nose tools.....31, 32, 36
for brass.....182, 183
hand tools.....166, 167
Round-nose tools, grinding.....37, 38
Rubber or fiber, machining hard..... 187
Rules, shrink..... 349
standard steel.....45, 305
Running fits.....68, 69
Running fits, schedule of operations for turning and filing.....98, 99
Rust on machines, to prevent..... 13

S

Schedule of operations for centering, squaring and straight turning.....57–59
Schedule of operations for chucking with a flat drill and chucking reamer in an engine lathe.....137, 138
Schedule of operations for cutting double Square thread..... 336
Schedule of operations for drilling and tapping a cross-feed screw nut in axial alinement in a milling machine saddle and knee.....337, 338
Schedule of operations for drilling, reaming and tapping.....246, 247
Schedule of operations for grinding forcing fits.....212, 213
Schedule of operations for grinding running fits.....210, 211
Schedule of operations for grinding standard mandrel.....213–215
Schedule of operations for laying out and turning eccentric shaft.....338, 339
Schedule of operations for laying out two-throw 90° crankshaft center fixtures.....340, 341
Schedule of operations for making brass binding post.....183–187
Schedule of operations for making brass nurled thumb nuts.....185–187
Schedule of operations for making cast-iron flange.....154–156
Schedule of operations for making duplicate pieces..... 54
Schedule of operations for making finished bolt and nut.....124, 125
Schedule of operations for making machine handle.....168–170
Schedule of operations for making pulley.....158–160
Schedule of operations for making Square-thread nut.....323–325
Schedule of operations for making Square-thread screw.....320–322
Schedule of operations for making standard mandrel.....149–152
Schedule of operations for making Twenty-nine degree thread screw..331–333
Schedule of operations for making Twenty-nine degree thread nut..333–335
Schedule of operations for straight turning and fitting.....85, 86

Page

Schedule of operations for turning and filing drive fits. *See* Shaft blank.....98, 99, 100
Schedule of operations for turning and filing running fits.....98, 99
Schedule of operations for turning and fitting a taper.....82, 83
Schedule of operations for turning crankshaft.....342, 343
Schedule of operations for turning engine eccentric.....339, 340
Schedule of operations for United States standard or Sharp V inside threading in the engine lathe...311, 312
Schedule of operations for United States standard or Sharp V threading in the engine lathe.....112–114
Schedule of operations for preparing two shaft blanks for fitting, multiple.....98, 99
Scraping fillets of cast iron (lathe work).....154, 155
radial faces of cast iron (lathe work)..... 153
Screw bolts, machine..... 293
Screw cutting. *See* Threading.
Screw cutting in the engine lathe, 110–119, 124, 125
on taper work..... 119
with rapid change-gear mechanism..... 8, 9
Screw cutting, catching the thread in 118
calculating simple gearing for.....110, 111
description of mechanism for.....111–113
lathe tools for, 95, 107–109, 310–320, 328–330
operating lathe for..112, 113
preparation of blank and nut for..... 111
setting United States standard or Sharp V tool for..... 108
Screw cutting, theory of.....110, 111
Screw cutting in an engine lathe, inside United States standard or Sharp V.....310–314
Screw cutting tools, inside United States standard or Sharp V, 95, 310, 311
Screw cutting tools, to set inside United States standard or Sharp V. 311
Screw dies, machine.....294, 295
gage, American Screw Company's machine..... 298
Screw heads, counterboring for...299–303
countersinking for..302, 303
Screw heads and bolts, spot facing for..... 300
Screw thread calipers..... 104
Screw thread to a shoulder in an engine lathe, cutting an inside.... 314
Screw threads. *See* Threads. 102–125, 269–299, 310–338, 345–349
per inch.....104, 105
Screw threads, calculating compound gearing for.....115, 116
calculating gearing to cut fractional....114–117

Digitized by Google

Page

Screw threads, calculating simple gearing for 110–113
counting 104, 105, 113
fractional 114–117
French standard 345
International standard 345
multiple 335, 336
pitch of 104
right and left 103
rolled 103
Sharp V .. 102–125, 310–314
Square 314–325
Twenty-nine degree. 325–335
United States standard 102, 106–109, 111–113, 124, 125
uses of different 103
Whitworth (English) standard ... 102, 119, 120
Screw threads with an English lead screw, calculating gearing for cutting metric 117
Screw threaded work, root diameter of 102, 104
Screws and nuts, automobile standard 286, 346, 347
methods of threading 103
Screws, cap 287, 288, 300
collar cap 288
examples of drilling, counterboring, and countersinking for machine or cap 300
fillister-head 288, 289, 293, 300
flat-head 288, 293, 298, 300
hexagonal-head 287, 288, 300
lag 289, 290
lead 2, 3, 6, 7, 103
machine 292, 293, 295–299, 363
multiple-thread 335, 336
nurled-head. 183–186, 188, 189, 292
oval countersunk head, 288, 293, 300
round-head 288, 293, 300
set 291, 292
Sharp V thread .. 102–125, 310–314
Square-head 248, 288
thread 314–325
thumb 292
Twenty-nine degree thread, 325-335
United States standard thread, 102, 106, 107, 109, 111–113, 124, 125
wood 290, 291
Screws and stud bolts, drilling for, 248, 249, 260–269, 300
Seasoning of ground work 208
Semi-finished bolts and nuts, 124, 125, 283, 284, 348, 349
Set-overs for turning tapers, table of footstock 360–362
Set screw chucks 232, 255
Set screws, cone-point 291, 292
cup-point 291, 292
drilling for ... 158–160, 247–250
hanger pivot-point 291, 292
headless 291, 292
round-point 291, 292
uses of 291, 292
Set screws for pulleys, locating ... 158–160
Setting dead center in alinement 18, 19
inside calipers 305
Square threading tool .. 319

Page

Setting inside Twenty-nine degree threading tool 330
United States standard and Sharp V screw threading tools 311
outside calipers 45
Square threading tools 318
studs by hand 284
power 266, 267
Twenty-nine degree threading tools 329
United States standard or Sharp V tools for threading 108
Shaft, polishing 162
grinding a slender 217, 218
turning a slender 192, 193, 194
Shaft blanks for grinding fits, schedule of operations for preparing 98, 99
Shaft blanks for turning and filing fits, schedule of operations for preparing 99–101
Shafting, rods and bolts, straightening 26
Sharpening dies 280
taps 271
Shear tools 92
Sheet and plate, steel and iron gage sizes, United States standard 371
Sheet metal, grinding twist drill for ... 232
twist drills for 232, 233
Shoulders, squaring 34, 121, 309, 310
Shrink rules 349
Shrinking fits 73, 75
Shrinkage of castings 349
Side tool, left 92
tools 29, 32–34, 87, 88, 92, 94, 95
tools, grinding 38–40
Silicate grinding wheels 200
Sizes of taps 269
Slender shaft, grinding 217, 218
Slide rest for speed lathe 179
Sliding fits 70
Sliding fits, table of 70
Slotting by drilling 253
Soap mixture for cutting tools 51
Soda water for cutting tools 51
Soft hammers for driving mandrel ... 148
Solid jig 260, 269
Speed for lathe filing 59, 60
of work for grinding 203
Speed indicator, surface speed attachment for 48, 49
Speed, number of revolutions to obtain surface 366, 367
Speed lathe 11, 12
countershaft 11, 12
Speed lathe, chucking in 170, 171
drilling in 255–257
methods of holding work in 164
slide rest for 179
spindle speeds for 239–241
tapping in 171, 172
Tee-rest for 11, 12, 164
Speeds of drilling machines, 222–224, 239–241
gear-driven vertical drilling machine 241
high-speed steel drills 239
Speeds for grinding wheels 202
hand tools, cutting 164
lathe tools, cutting 47–50
Sperm oil 50
Split pin 286, 346

Digitized by Google

Page

Spot facing for heads of bolts and screws 300
Spring calipers 45–47
chuck 130, 294, 295
cotter 286, 346, 347
tools 91, 92
winding in engine lathe 179–181
Spring, extension 179
Spring winder, winding springs with 180
Springs, coil 179, 181
compression 179
Square dog 121
Square-head bolt 248, 283
cap screws 248, 288
Square-nose tools 37
Square, calculating diameter to turn blank for milling or filing 303
Squaring to shoulder 34, 121, 309, 310
with hand tools 166
with inside squaring tool, . 309, 310
with graver 166
Squaring, removing burr when 33, 34
rough 33
step method of 87, 88, 166
two forged tools for 96
Squaring cast iron, round-nose tool for 31
Squaring cast iron, side tools for, 29, 32–34, 87, 94, 95
Squaring steel, side tools for, 87, 88, 92, 94, 95
Square thread taps 320
Square thread nut, fitting screw to .. 325
schedule of operations for making 323–325
Square thread screw, schedule of operations for making a 320–322
Square thread screw and nut, sectional view of 315
Square threading tool angle gage 317
holders and cutters 95, 318
Square threading tool, setting inside . 319
setting 318
Square threading tools 29, 95, 315–318
Square threading tools, inside 29, 95, 319, 320
grinding 40, 41
Square threads 314–325
Square threads, formulas for 315, 316
width and inclination of tool for multiple 336
Squares, caliper 66, 67
Standard cylindrical plug and ring gages 77
Standard and limit gages 77–79
holes 75, 76
solid mandrels or arbors, 146, 349–352, 364, 365
Steady rests 191–193
Steady rests, turning shaft supported with 192
Step method of squaring 87, 88, 166
with hand tools in speed lathe 166
Steel wire 368–370, 372
Steel, twist drills for drilling hard . 231, 232
Steel rules, standard 45, 305
Steel or wrought iron, diamond-point tool for finish turning 90–92
Steel or wrought iron, diamond-point tool for rough turning 88

Page

Steel or wrought iron, drilling 230, 231
finish squaring, 87, 88, 96
turning, 90, 91, 92
lathe tools for, 29, 87–98
rough squaring, 87, 88
turning, 88–90
Stock, cutting off 97, 98
Stock (material), inspection and measurement of 52
Straight-groove drills 233
shank twist drills 230
Straight grinding, setting swivel table for 206
Straight turning and fitting, schedule of operations for 85, 86
Straightening shafting, rods and bolts 26
Stubs' steel wire gage sizes 369, 372
iron wire gage sizes 368, 372
Stud bolts 266, 267, 283, 284
holder 284
Studs for driving large work 149
Studs by hand, setting 284
power, setting 266, 267
Stud bolts and screws, drilling for, 248, 249, 264–267, 268, 284
Surface speed attachment for speed indicator 48, 49
Surface speed, table of number of revolutions required to obtain 366, 367
Swing of lathes 7

T

Table and formulas for bolt heads and nuts 303, 348, 349
[illegible]ble center 256
of allowances and limits for standard holes 76
automobile standard screws and nuts 346, 347
Brown & Sharpe tapers .. 356, 357
center-hole dimensions 21
common fractions and decimal equivalents 64
decimal equivalents of different steel music wire gage sizes 370
decimal equivalents of Stubs' steel wire 369
driving fits 70
estimated weight of core prints and cores for castings 349
forcing fits 71
footstock set-overs for turning tapers 360–362
grades of grinding wheels . 198, 201
International and French standard threads 345
Jarno tapers 358, 359
lathe cutting speeds 49
limits of hole diameters 76
lubricants for cutting tools. 51
Morse tapers 354, 355
number of revolutions required to obtain surface speed 366, 367
Sharp V screw threads 106
shrinkage of castings 349
sliding fits 70

Digitized by Google

Page
Table of speeds for grinding wheels.. 202
and feeds for carbon steel drills.....237–239
speeds and feeds for high-speed steel drills..... 239
spindle speeds for belt-driven vertical drilling machine 239–241
spindle speeds for speed lathe.....239–241
spindle speeds for a high-speed vertical drilling machine..... 241
Square threads 316
standard mandrel dimensions 364, 365
thread parts of Twenty-nine degree threads..... 327
Twenty-nine degree threads 328
United States standard bolts and nuts..... 348
United States standard Sharp V thread taps and tap drill sizes.....278, 279
United States standard sheet and plate steel and iron gage size..... 371
United States standard screw threads..... 107
weight of castings from wooden patterns..... 349
Whitworth (English) standard threads..... 120
Tables of A. S. M. E. standard and special machine screws, taps, and tap drill sizes.....296, 297
decimal equivalents of American Screw Company's machine screws, taps, and tap drill sizes.....298, 363
different drill sizes, consecutive and comparative.....350–353
different wire gages in general use, comparative... 368
gage sizes of twist drills..234–236
running fits.....68, 69
shrinking fits..... 75
Tanite grinding wheels.....198, 200
Tap bolts..... 288
drill diameters (how obtained).. 278
drills..... 237
holes.....272, 273
wrenches.....271–274, 277
Tap, interrupted thread..... 313
to finish thread backing plate with.....312–314
Tap cut large, making a..... 276
Tap wrench, single-end..... 272
Taper forcing fits.....71–74
gage..... 82
pin.....144, 145
running fits..... 68
shank twist drills.....230–232
turning.....80–86
with foot stock set-over.....81, 83
with taper attachment.....84, 85
Taper, schedule of operations for turning and fitting a.....82, 83
Taper attachment, using 84, 85
bushing, grinding..... 221
collet, grinding..... 220

Page
Taper grinding, setting swivel table for..... 206
Taper holes in a speed lathe by power, reaming.....144, 145
Taper pin reamers, fluted.....144, 145
Taper work, setting tool to thread... 119
Tapering or crowning pulleys.....156–159
Tapers, Brown & Sharpe.....356, 357
calculating distance to set over footstock for.....81, 82
grinding.....213–215, 219–221
Jarno.....358, 359
methods of grinding..... 206
Morse.....354, 355
standard and special..... 80
table of footstock set-overs for turning.....360–362
use of patterns to obtain.... 82
Tapers are expressed, how..... 80
Tapped holes, 171, 172, 236, 237, 246–248, 273–279, 296–299
Tapped holes, depth of.....272, 273
Tapping attachment.....266, 267
backing plates of chucks..312, 313
by hand in vertical drilling machine.....274, 275
for cap screws.....247, 249
in engine lathe.....312, 313
in speed lathe.....171, 172
loose nuts.....272, 273
with machine screw tap.... 294
Tapping, accurate hand.....274, 275
alinement drilling and...337, 338
approximate.....273, 274
hand.....269–279, 293, 294
jigs for.....265, 275
power.....266, 267, 277
Taps.....269–279, 293, 294
Taps, adjustable.....271, 313
automatic..... 271
clearance of..... 271
lands of..... 270
master..... 271
machine-screw.....293, 294
multiple-threaded..... 336
nut..... 271
pulley..... 271
removing broken 276, 277
set of hand..... 270
sharpening..... 271
sizes of..... 269
Square thread 320
threading pipe..... 119
Twenty-nine degree thread ... 330
Taps and tap drill sizes of American Screw Company's machine screws. 298
Taps and tap drill sizes of A. S. M. E. standard machine screws 296, 297
Taps and tap drill sizes of Sharp-V threads.....278, 279, 299
Taps and tap drill sizes of United States standard threads.....278, 279
Templets.....167, 168
Ten-thousandth micrometer caliper, 67
Testing angle of centers..... 15
axial truth of center punch marks with indicator...133, 134
Thread calipers..... 104
Thread, cutting Square.....314–325
cutting Sharp V..110–117, 310–314
cutting United States standard.....110–117, 310–314

Digitized by Google

Page

Thread, gage for Sharp V..... 108
lead of.....104, 117
Twenty-nine degree.....325–335
Thread gage, grinding tools to fit United States standard..... 109
Thread screw to nut, fitting Square . 325
fitting Twenty-nine degree, 332, 333
Thread screw to nut, fitting United States standard or Sharp V.... 114, 125
Thread to a shoulder in an engine lathe, cutting an inside screw..... 314
Threaded work, diameter of..102, 104, 111
root diameter of..102, 104
Threading, 102–120, 124, 125, 172–174, 279–281, 294, 295, 326–336
catching the thread in 118
dies.....279–281, 294,
in engine lathe, 110–119, 124, 125
in speed lathe.....172, 173
nuts, 102–120, 173, 174, 183, 185, 186, 269–279, 292–294, 310, 338
taper work..... 119
tool gage for grinding and setting Twenty-nine degree thread tool 329, 330
tool holders and cutters, 94, 95, 318
tools for United States standard or Sharp V thread.....29, 95, 107–109
with machine screw dies, 294, 295
rapid change-gear mechanism 8, 9
Threading, calculating compound gearing for.....115, 116
calculating simple gearing for.....110, 111
description of mechanism for.....111–113
hand.....279–281, 294, 295
lathe tools for, 95, 107–109, 310–320, 328–330
operating lathe for...112, 113
power..... 280
preparation of blank and nut for..... 111
setting United States standard or Sharp V tool for..... 108
theory of.....110, 111
Threading in the engine lathe, inside, United States standard or Sharp V, 310–314
Threading to a shoulder, inside..... 314
Threading tool, angle gage for Square 317
Threading tools, 29, 94, 95, 107–109, 310–320, 328–330
Threading tools, inside Square 29, 95, 319, 320
inside United States standard or Sharp V.....95, 310, 311
making Square 317
multiple..... 336
setting inside United States standard or Sharp V 311
setting inside Square 319
setting inside Twenty-nine degree... 330

Page

Threading inside setting United States standard or Sharp V..... 108
Square....29.95, 315–318
Twenty-nine degree, 29, 95, 326, 328, 329
United States standard or Sharp V, 29, 94, 95, 107–109
United States standard or Sharp V inside screw..95, 310, 311
Whitworth (English) standard 120
Threads, 102–125, 269–299, 310–338, 345–349
per inch.....104, 105
Threads, British Association standard, 102
calculating compound gearing for.....115, 116
calculating gearing for given lead of.....104, 117
calculating gearing to cut fractional.....114–117
counting.....104, 105, 113
double.....335, 336
fractional.....114–117
French standard..... 345
International standard 345
left.....114, 314
metric.....102, 117, 345
multiple.....335, 336
pitch of..... 104
right and left..... 103
rolled..... 103
screw, 102–125, 269–299, 310–338, 345–349
Sharp V..... 102–125, 310–314
Square..... 314–325
Twenty-nine degree.....325–335
United States standard, 102, 106, 109, 111–113, 124, 125, 310–314
uses of different..... 103
Whitworth (English) standard.....102, 119, 120
Threads with an English lead screw, calculating gearing for cutting metric screw..... 117
Three-groove twist drills for drilling cored holes.....250, 251
Thumb nuts..... 292
screws..... 292
Thumb nuts, schedule of operations for making brass nurled.....185–187
Tool holders and cutters, cutting off 93–95, 98
Tool holders and cutters, threading, 94, 95, 318
Tool holders, boring.....95, 310
double..... 96
Tool holders and cutters 93–96, 98, 310–312, 318
Tools, boring.....29, 95, 308–310
chart of forged lathe.....28, 29
chattering of..... 91
forming.....29, 95, 190, 191
half-diamond point.....29, 93
hand.....166, 167
height of lathe cutting..27–37, 87–97
high-speed steel lathe, 28, 90, 91, 93–95
large roughing.....29, 89, 90
lathe.....27–44, 87–98

Digitized by Google

Page
Tools, lathe box for....53, 55
lathe cutting-off...29, 94, 95, 97, 98
lathe threading, 29, 94, 95, 107–109, 310–320, 330–338
left diamond-point.... 93
nurling....188, 189
oilstoning.... 44
rake, clearance and cutting angles of lathe cutting....27, 28
right and left, lathe.... 30
round-nose....31, 32, 36
shear.... 92
side....29, 32–34, 87, 94, 95
small finishing....90, 91
roughing....29, 89
spring....91, 92
square-nose.... 37
Square threading... 29, 95, 315–318
theory of cutting....27, 28
Twenty-nine degree threading, 29, 95, 326, 328, 329
Tools for cast iron, diamond-point...34–37
lathe....27–37
Tools for steel or wrought iron, diamond-point....88, 90, 91
Tools for steel or wrought iron, lathe....29, 87–98
Tools made from old files, hand.... 178
Tools set at different heights, effect of lathe....30, 35, 36
Tee rest for speed lathe....11, 12, 164
Truing engine-lathe centers....16, 17
grinding machine centers..204, 205
wheels....203, 204
speed lathe centers.... 18
Tube holes, cutter for drilling boiler, 254, 255
Turnbuckles.... 286
Turn to setting of outside calipers, adjusting tools to.... 46
Turning an engine eccentric....339, 340
a rod held in draw-in-chuck 130
a slender shaft....192, 193, 194
brass with a planisher....175, 176
bushing held in a chuck.... 128
crankshaft....342–343
steel or wrought iron with large finishing tool....90, 91
steel or wrought iron with large roughing tool.... 90
steel or wrought iron with small roughing tool.... 89
steel or wrought iron with small finishing tool....90, 91
taper....80–86
to a desired diameter.... 46
with graver....164–167
work of one diameter from end to end....56, 57
Turning, curve....190, 191
eccentric....338–343
expansion of work while... 56
finish.... 28
hand....164–177, 178
order of rough and finish... 52
rough.... 28
shaft....338, 339
Turning brass with a ripper, rough 174, 175
Turning cast iron, diamond-point tool for rough.... 36
round-nose tool for finish....36, 37
Turning cast iron, round-nose tool for rough.... 31
Turning steel or wrought iron, diamond-point tool for finish....90, 91
Turning steel or wrought iron, diamond-point tool for rough.... 88
Turning steel or wrought iron, small finishing tool for.... 90
Turning with taper attachment....84, 85
Turpentine for machining aluminium or glass....51, 52
Twenty-nine degree thread taps.... 330
Twenty-nine degree thread nut, schedule of operations for making...333–335
Twenty-nine degree thread screw and nut, sectional view of.... 325
Twenty-nine degree thread screw, schedule of operations for making....331–333
Twenty-nine degree threading tool gage....329, 330
Twenty-nine degree threading tools, 29, 95, 326, 328, 329
Twenty-nine degree threading tools, setting.... 329
Twenty-nine degree threading tools, grinding....328–330
Twenty-nine degree inside threading tools, setting.... 330
Twenty-nine degree threading tools, inside....29, 95, 329, 330
Twenty-nine degree threads....325–335
Twenty-nine degree threads, formulas for.... 326
Twist drill gages....234–236
grinder....227, 228
grinding gage.... 229
Twist drills....225–259
for brass....232, 233
cast iron 230–232, 234–237, 250, 251
hard steel....231, 232
steel and wrought iron, 230–232, 234–237, 257–259
thin sheet metal...232, 233
made by twisting from flat bar....231, 232
Twist drills, chucking with....135, 171
examples of good and bad grinding....225, 226
gage sizes of 234–236, 350–353
hand method of grinding, 228, 229
machine method of grinding....227, 228
regular sizes of....234–237
straight-shank.... 230
taper-shank....230–232
Twist drills for brass, grinding.... 232
Twist drills for thin sheet metal, grinding.... 232

U

United States standard threading tools, grinding....41, 42, 109
United States standard threads, 102, 106–109, 111–113, 124, 125, 310–314
United States standard thread gage, 109
or Sharp V inside threading tools 29, 95, 310, 311
United States standard or Sharp V threading tools....29, 94, 95, 107–109

Digitized by Google

Page

United States standard screw, sectional view of 102, 107
Units, electrical 344
Universal chucks 128
grinding machine 196, 197
tool grinder 43, 44
Universal chucks, truing and holding work in 128
Universal grinding machine, operating 209

V

Vaseline 13
Vernier calipers 65–67
principle 65
Vernier calipers, measuring with 66
reading 65, 66
reading inside 66
Vertical drilling machine described . 222–224
Vertical drilling machine, spindle speeds for high-speed 241
spindle speeds for belt-driven . . 239–241
Vertical drilling machines, hand and power feed for 222–224, 240, 241
Vise, jig 267, 268
machinists' 281
Vitrified grinding wheels 198, 199, 201
Volt 344
Vulcanite grinding wheels 198, 200
Vulcanite, machining 187
V blocks 251
center 256
V thread, formulas for Sharp 105
V thread gage, Sharp 15, 108
screw, sectional views of Sharp 102, 105
threading tools, grinding 41, 42
V threads, Sharp 102–125, 310–314

Page

W

Washers 285, 286, 290
Water for cutting tools, soda 51
grinding 206, 207
guards 207, 209, 210
Watt 344
Ways of a machine, oiling 13
Weight of castings from wooden patterns 349
Weight of core prints and cores for castings, estimated 349
Wet tool grinder 42–44
Wheels, grinding 198–202
Whitworth (English) standard threading tools 120
threads 102, 119, 120
Whitworth thread screw, sectional view of 119
Winding springs 179–181
Wing nuts 292
Wire gages 234–236
Wire, brass 368, 372
copper 368, 372
cutting off 177, 178
iron 368, 372
steel 368–370, 372
Wire and plate gages, uses of different 372
Wire gages in general use, comparative tables of 368
Wrenches, tap 271–274, 277
Wrought iron, lathe tools for steel or, 29, 87–98
Wood screws 290, 291
Wood, drilling 232
fastening metal to 291
Work, driving lathe 20, 21

Z

Zero lines 18, 20

Digitized by Google

Digitized by Google

Digitized by Google

Digitized by Google

89094998986

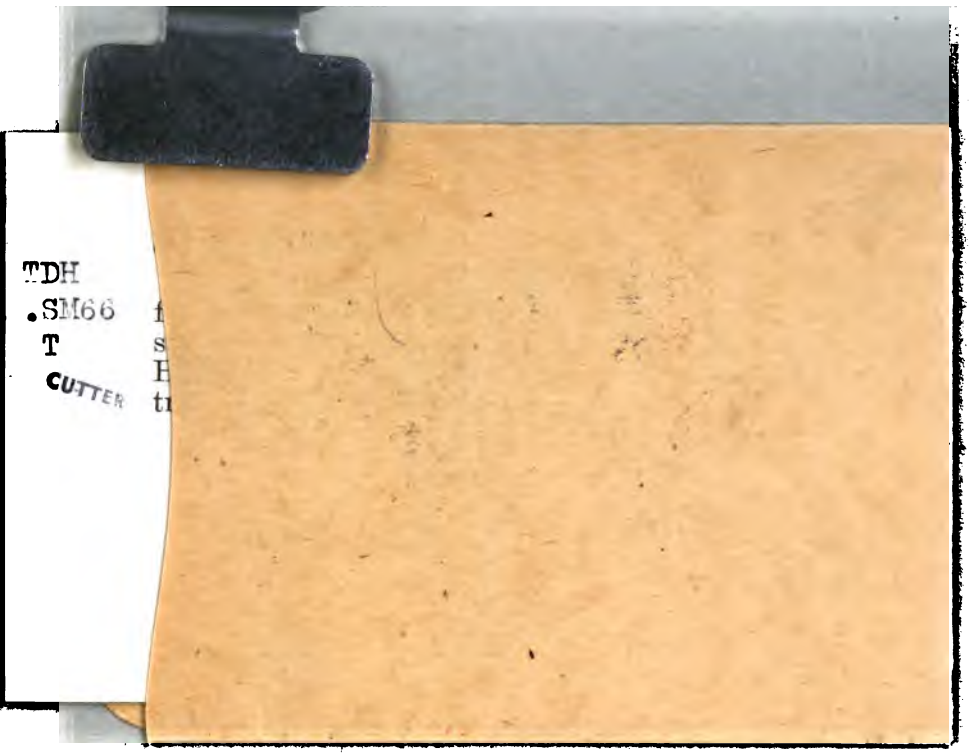
b89094998986a

y be kept

A fine

the bo

Digitized by Google


TDH
•SM66
T
CUTTER

Google

B89094998986A